Watershed Modeling System

WMS v7.1

TUTORIALS

WMS 7.1 Tutorials

Copyright © 2004 Brigham Young University - Environmental Modeling Research Laboratory

All Rights Reserved

Unauthorized duplication of the *WMS* software or documentation is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING RESEARCH LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Environmental Modeling Research Laboratory of Brigham Young University. For more information about this software and related products, contact the EMRL at:

Environmental Modeling Research Laboratory
Rm. 242, Clyde Building
Brigham Young University
Provo, Utah 84602
Tel.: (801) 422-2812
e-mail: wms@byu.edu
WWW: http://www.emrl.byu.edu/wms.html

For technical support, contact your *WMS* reseller.

TABLE OF CONTENTS

1 INTRODUCTION 1-1

1.1 SUGGESTED ORDER OF COMPLETION 1-1
1.2 TUTORIAL FILES 1-2
1.3 STARTING OVER 1-2
1.4 GETTING AROUND THE WMS INTERFACE 1-2

2 IMAGES 2-1

2.1 OBJECTIVES 2-1
2.2 GEOTIFF FILES 2-2
2.3 WORLD FILES 2-4
2.4 REGISTERING SCANNED IMAGES 2-4
2.5 AERIAL PHOTOGRAPHS 2-7
2.6 CONCLUSION 2-10

3 BASIC FEATURE OBJECTS 3-1

3.1 OBJECTIVES 3-1
3.2 CREATING AND EDITING FEATURE OBJECTS 3-2
3.3 USING SHAPE FILES TO CREATE FEATURE OBJECTS 3-6
3.4 CREATING FEATURE OBJECTS USING BACKGROUND IMAGES 3-8
3.5 MORE FEATURE OBJECTS FROM IMAGES 3-11
3.6 DISPLAY OPTIONS 3-12
3.7 CONCLUSION 3-13

4 DEM BASICS 4-1

4.1 OBJECTIVES 4-1
4.2 GETTING DEMS FROM THE INTERNET 4-2
4.3 MERGING DEMS 4-7
4.4 TRIMMING DEMS 4-8
4.5 DISPLAYING DEMS 4-9
4.6 CONCLUSION 4-11

5 USING TRIANGULATED IRREGULAR NETWORKS 5-1

5.1 OBJECTIVES 5-1
5.2 IMPORTING SURVEY DATA 5-2
5.3 DIGITIZING DATA 5-2
5.4 TRIANGULATION 5-3
5.5 AUTOMATED TIN EDITING 5-3
5.6 MANUAL TIN EDITING 5-6
5.7 CREATING A TIN USING A CONCEPTUAL MODEL 5-12
5.8 CONVERT TO DEM 5-16
5.9 EXPORTING DATA TO CAD 5-17

6 ADVANCED FEATURE OBJECTS 6-1

6.1 OBJECTIVES 6-1

6.2 Creating a Watershed from Scratch with Feature Objects 6-2
6.3 Cleaning 6-5
6.4 Feature Objects from CAD Data 6-13
6.5 Conclusions 6-15

7 DEM DELINEATION 7-1

7.1 Objectives 7-1
7.2 Importing DEM Data 7-2
7.3 Computing Flow Data and Accumulations 7-3
7.4 Delineating Watersheds from DEMs 7-4
7.5 Creating Sub-Basins 7-6
7.6 Adding a Stream Arc and Redefining Basins 7-9
7.7 Displaying DEMs 7-13
7.8 Conclusions 7-15

8 EDITING DEMS 8-1

8.1 Objectives 8-1
8.2 Running TOPAZ and Basin Delineation 8-2
8.3 DEM Fill Command 8-3
8.4 Editing Flow Directions 8-6
8.5 Editing Elevations to Create Streams 8-11
8.6 Editing Elevations Using Feature Arcs 8-16
8.7 Computing a Storage Capacity Curve 8-19
8.8 Hydrograph Routing 8-19

9 TIME OF CONCENTRATION CALCULATIONS AND COMPUTING A COMPOSITE CN 9-1

9.1 Reading a TIN File 9-2
9.2 Defining Flow Path Arcs 9-3
9.3 Assigning Equations to Time Computation Arcs 9-6
9.4 Using the Time Computation Arcs to Compute Time of Concentration for a TR-55 Simulation 9-8
9.5 Using the Time Computation Arcs to Compute the Travel Time Between Outlet Points 9-11
9.6 Computing a Composite Curve Number 9-11
9.7 More TR-55 9-14
9.8 Conclusions 9-15

10 HEC-1 INTERFACE 10-1

10.1 Objectives 10-1
10.2 Delineating the Watershed 10-2
10.3 Single Basin Analysis 10-7
10.4 Computing the CN Using Land Use and Soils Data 10-11
10.5 Adding Sub-basins and Routing 10-12
10.6 Modeling a Reservoir in HEC-1 10-18
10.7 Reviewing Output 10-22
10.8 Conclusion 10-22

11 RATIONAL METHOD INTERFACE 11-1

11.1 Reading in Terrain Data 11-1
11.2 Running a Rational Method Simulation 11-2
11.3 Adding a Detention Basin 11-10
11.4 Conclusions 11-11

12 NATIONAL FLOOD FREQUENCY PROGRAM (NFF) INTERFACE 12-1

12.1 OPENING THE DRAINAGE BASIN 12-1
12.2 PREPARE THE BASIN FOR USE WITH NFF 12-2
12.3 CALCULATING PERCENTAGE OF LAKE COVER 12-3
12.4 RUNNING NFF 12-6
12.5 UTILIZING AN NFF REGION COVERAGE 12-8
12.6 CONCLUSIONS 12-12

13 HEC-RAS ANALYSIS 13-1

13.1 OBJECTIVES 13-1
13.2 PREPARING THE CONCEPTUAL MODEL 13-1
13.3 CREATING THE NETWORK SCHEMATIC 13-10
13.4 CREATING THE GEOMETRY IMPORT FILE 13-12
13.5 USING HEC-RAS 13-12
13.6 POST-PROCESSING 13-15

14 HEC-RAS – MANAGING CROSS SECTIONS 14-1

14.1 OBJECTIVES 14-1
14.2 CREATING A CONCEPTUAL RIVER MODEL 14-2
14.3 CONVERTING A DEM TO A TIN 14-4
14.4 EXTRACTING CROSS SECTIONS 14-5
14.5 MERGING CROSS SECTIONS 14-5
14.6 RUNNING HEC-RAS 14-11
14.7 FLOODPLAIN DELINEATION 14-14

15 FLOODPLAIN DELINEATION 15-1

15.1 OBJECTIVES 15-1
15.2 FLOODPLAIN DELINEATION OPTIONS 15-2
15.3 CREATING A SCATTER POINT FILE 15-4
15.4 CREATING SCATTER POINTS WITH THE CHANNEL CALCULATOR 15-6
15.5 DELINEATION FROM HEC-RAS DATA 15-12
15.6 CREATING A FLOOD EXTENT COVERAGE 15-17
15.7 CREATING A FLOOD DEPTH COVERAGE 15-18
15.8 CONCLUSIONS 15-18

16 STOCHASTIC MODELING USING HEC-1 AND HEC-RAS 16-1

16.1 OBJECTIVES 16-1
16.2 OPENING THE HEC-1 AND HEC-RAS MODELS 16-2
16.3 RUNNING THE STOCHASTIC MODEL 16-4
16.4 VIEWING THE RESULTS 16-7

17 SIMPLIFIED DAM BREAK 17-1

17.1 PREPARING THE MODEL 17-1
17.2 USING SMPDBK 17-7
17.3 POST-PROCESSING 17-9

18 STORM DRAIN: RATIONAL DESIGN 18-1

18.1 OBJECTIVES 18-1
18.2 DEFINING RUNOFF COEFFICIENTS 18-2
18.3 DEFINING THE DRAINAGE AREA 18-3
18.4 IMPORTING THE PIPE NETWORK 18-7
18.5 LINKING NODES AND ASSIGNING ELEVATIONS 18-12
18.6 SAVING THE SIMULATION AND RUNNING STORM DRAIN 18-13

19 STORM DRAIN: HYDROGRAPHIC DESIGN 19-1

19.1 OBJECTIVES 19-2
19.2 DEVELOPING THE SURFACE DRAINAGE COVERAGE 19-2
19.3 RUNNING A RATIONAL ANALYSIS 19-6
19.4 CREATING THE PIPE NETWORK 19-8
19.5 SAVING THE SIMULATION AND RUNNING STORM DRAIN 19-13

20 HSPF INTERFACE 20-1

20.1 OPENING THE WATERSHED AND INITIALIZING THE HSPF MODEL 20-1
20.2 IMPORTING LAND USE AND SEGMENTING THE WATERSHED 20-3
20.3 AGGREGATING SEGMENTS 20-5
20.4 DEFINING LAND SEGMENT PARAMETERS 20-6
20.5 DEFINING REACH SEGMENT PARAMETERS 20-10
20.6 CREATING MASS LINKS 20-12
20.7 SAVING AND RUNNING HSPF SIMULATION 20-13

21 CE-QUAL-W2 INTERFACE 21-1

21.1 OBJECTIVES 21-1
21.2 IMPORTING DATA AND COORDINATE CONVERSION 21-2
21.3 DETERMINING RESERVOIR EXTENT 21-3
21.4 CHANGE CONTOUR DISPLAY OPTIONS 21-5
21.5 CREATE BRANCHES 21-5
21.6 CREATE SEGMENTS 21-7
21.7 SYSTEM MODELING 21-9
21.8 SAVING BATHYMETRY AS CE-QUAL-W2 INPUT FILE 21-13

CHAPTER 1

Introduction

This document contains tutorials for the Watershed Modeling System (*WMS*). Each tutorial provides training on a specific component of *WMS*. Since the *WMS* interface contains a large number of options and commands, you are strongly encouraged to complete the tutorials before attempting to use *WMS* on a routine basis.

In addition to this document, the *WMS Help File* also describes the *WMS* interface. Typically, the most effective approach to learning *WMS* is to complete the tutorials and use the help file when encountering a portion of *WMS* that is unclear.

1.1 Suggested Order of Completion

This first tutorial allows you to open the interface and get familiar with the WMS Interface in general. Chapters 2-8 cover the basic data structures used in WMS (images, GIS vector data or feature objects, DEMs, and TINs) and how watersheds can be delineated to set up hydrologic models from them. Beginning with chapter 9 the various interfaces to hydrologic and hydraulic models are covered and you can choose (see the table of contents) from among them which ones you wish to work on first.

The tutorial for the public domain version of WMS and for the 2-D modeling capabilities are treated in separate documents.

1.2 Tutorial Files

Each tutorial has one or more files that have been prepared for you to use. You are instructed at various points to open these files. The default installation of WMS copies all of these files into a directory named "**tutorial**". Further, the files for each chapter are organized by directory within the tutorial directory. For example, if you install WMS at "**C:\WMS71**" then all of the files for this first tutorial will be found in "**C:\WMS71\tutorial\ch1**", and all of the files for the second tutorial will be found in "**C:\WMS7\tutorial\ch2**", and so on for the remaining chapters.

1.3 Starting Over

It is suggested that you start WMS new at the beginning of each tutorial. If you continue from one tutorial to another without quitting, then data, display options, and other WMS settings may not be in sync with the tutorial instructions causing the exercise to become confusing.

1.4 Getting Around the WMS Interface

The WMS Help file has a section on some of the basic elements of the WMS graphical user interface (GUI). In this tutorial we will refer you to that information and let you take a self-guided tour in order to get familiar with GUI.

1.4.1 Quick Tour

If you haven't yet taken the quick tour, do so now. If you have then you can skip on to the next section.

1. Start WMS
2. Select ***Help | WMS Help***
3. Follow the links through the Quick Tour to completion
4. Close the Help file when you are finished

1.4.2 Self-Guided Tour

The WMS Help file contains some basic information for some of the important elements of the GUI. In this section you should review these help pages and then practice, on your own so that you understand how the interface works.

1. Start WMS if you are not continuing from the previous section

2. Select ***Help*** | ***WMS Help***

3. Select the link near the bottom of the opening page that says, "Getting around the WMS Interface."

4. Review these sections and then close the help file

5. Select ***File*** | ***Open***

6. Open the file named *trailmt.dem*

7. Select ***File*** | ***Open***

8. Open the file named *trailmt.tif*

As a minimum be sure that you are comfortable with the following operations within the WMS interface (if you have questions refer back to the Getting around the WMS Interface page within the WMS Help File):

- Switching Modules
- Switching tools
- Zooming, panning, framing the image, and rotating in 3D. When you are finished use the plan view macro to make sure you are back in plan view before trying other things.
- Using the display options, contour options, and other macros
- Turning objects on and off and accessing menus from the Data Tree window.
- Turning objects on and off and changing display options.
- Contour options.
- Saving a project file.

If time permits continue exploring the different elements of the interface and/or reviewing the information within the WMS Help file.

CHAPTER 2

Images

Images are an important part of most projects developed by WMS. An image is comprised of a number of pixels (picture elements), each with its own color. The resolution, or size, of the pixels will determine the amount of area and detail represented in the image. Images are used in WMS to derive data such as roads, streams, confluences, land use, soils, etc. as well as providing a base map or "backdrop" to your watershed. In order to make use of images they must be georeferenced. Georeferencing an image defines appropriate x and y coordinates so that distances and areas computed from the image will be accurate. Because images are commonly used in GIS programs like WMS, data developers often store the georeferencing information as either part of the image file (a geotiff file for example), or in a separate file commonly referred to as a "world" file.

2.1 Objectives

In this workshop you will learn the primary ways that images are imported and georeferenced, or registered by WMS.

1. Learn how to use geotiff files
2. Understand what resampling an image means
3. Learn how to use world files
4. Learn how to register scanned images

5. Learn what a WMS image file is and how to save one
6. Learn how to use aerial photographs

2.2 Geotiff Files

Geotiff images are files that store georeferencing information. This means that you do not have to specify coordinates when you read in the image – it is done for you automatically. Geotiff images are available in different resolutions. In this part of the tutorial you will open three different resolutions for a topographic map of the same area. In the first part you will actually see how you can tile multiple images together.

2.2.1 USGS 1:24000 Quadrangle Maps

1. Switch to the *Map* module
2. Select ***Images*** | ***Import…***
3. Open "*trailmountain.tif*". This is a 1:24000 resolution TIF image.

4. Right-click on the trailmountain.tif icon in the Data Tree window and choose the ***Crop Collar…*** command
5. Choose the *Zoom* tool
6. Single-click on the image to zoom in
7. Keep zooming in until the display of the image is clear
8. Select the Frame Image macro
9. Select ***Images*** | ***Import…***
10. Open "*marysvalecanyon.tif*". This is an adjacent 1:24000 map image.
11. Right-click on the marysvalecanyon.tif icon in the Data Tree window and choose the ***Crop Collar…*** command
12. Select ***Images*** | ***Import…***
13. Open "*josephspeak.tif*". This is an adjacent 1:24000 map image.
14. Right-click on the josephspeak.tif icon in the Data Tree window and choose the ***Crop Collar…*** command

15. Select ***Images*** | ***Import…***

16. Open "*redridge.tif*". This is an adjacent 1:24000 map image.

17. Right-click on the redridge.tif icon in the Data Tree window and choose the ***Crop Collar…*** command

18. Try zooming in and see if you can see where the map seams are (hopefully you will have some difficulty, but if you look close enough you may be able to tell)

19. Select the Frame Image macro

20. Turn off the display of all four images by selecting the check marks in the boxes to the left of each image file icon in the Data Tree window

21. Select ***Images*** | ***Import…***

22. Open "*richfield100.tif*"

23. Crop the collar of this image by right-clicking on its' icon in the Data Tree window and choosing ***Crop Collar…***

This is a 1:100,000 resolution TIF image for the same area as *trailmountain.tif*

1. Zoom in on the image with the *Zoom* tool

2. Turn off the display of *richfield100.tif* from the Data Tree

3. Select ***Images*** | ***Import…***

4. Open "*richfield250a.tif*"

5. Crop the collar of this image by right-clicking on its' icon in the Data Tree window and choosing ***Crop Collar…***

This is a 1:250000 resolution TIF image for the same area as *trailmountain.tif*

1. Zoom in on the image with the *Zoom* tool

When you zoomed in on the three images, you may have noticed that as the map scale increased, the map showed less detail. 1:24000 maps cover far less area than 1:100000 or 1:250000 maps, so they can show much more detail. It would take thirty-two 1:24000 maps to cover the same area that is covered by one 1:100000 map. If you need a great amount of detail for your watershed, you may want to use the 1:24000 maps. However, if your watershed is very large, this size of map will provide too much detail. It will be difficult to see 'the big picture' of your watershed.

2.2.2 Manual Resampling

You can see that each .tif image covers a different sized area. When you zoom in on a portion of the image, it is resampled, or redrawn. If you find that zooming in or panning is taking an excessively long time, you may want to change the resample option to manual. You can do this by selecting the Display menu and choosing the Manual Resample option. Then, when you want to resample the image, choose the Resample command from the Display menu.

2.3 World Files

Many .tif image files do not contain georeferencing information. Some organizations distribute world files containing the georeferencing information along with the .tif files. These world files usually have the same name as the corresponding .tif file, but with the extension .tfw (for jpeg files the extension is .jgw). If you download a world file and are asked to supply a name for it, follow this naming convention. Use the following procedure to open a .tif image and its corresponding georeferencing information in WMS:

1. Delete all of the images one at a time by selecting them and choosing the <Delete> key, or by right-clicking and choosing the ***Delete*** command.

2. Select ***Images*** | ***Import…***

3. Open "*richfield250b.tif*"

Because there is a world file named *richfield250b.tfw* the image is automatically registered. If a world file for a tiff image is not named with the .tfw (or for a jpeg it is not .jgw) then you would have the option of importing the world file from within the registration dialog.

2.4 Registering Scanned Images

Sometimes you will not be able to obtain a geotiff image or an image with a world file. In this case, you will need to register the image manually. To do this, you will need to know the coordinates of three points on the image. These coordinates can be in either the UTM system or the geographic system. Before you scan your paper image, you will want to mark the three points you have selected with a '+' so that you can easily find the points when you register the image in WMS.

We will use a part of a soils file as a "scanned image" that will be used later to develop a soils coverage and then later to compute composite curve number.

1. Delete *richfield250b.tif* from the Data Tree

2. Select ***Images*** | ***Import…***

3. Open "*soils.tif*"

An image representing soil types will appear in the 'Register Image' dialog. Three small red *x*'s are marked on the image (these are small and difficult to see). There are also three plus signs with the numbers 1, 2, and 3 by them (much larger and visible). You need to place the numbered plus signs over the small x's and enter the appropriate coordinates in order to register the image.

1. Use the point selection tool to drag each red + over the corresponding X as shown in Figure 2-1. You may wish to place them close and then zoom in on the area for more accuracy. Once you zoom in there is a frame tool that you can use to re-center the image so that you can zoom in on another registration point. Do your best to move each plus mark over the corresponding red x in the image now.

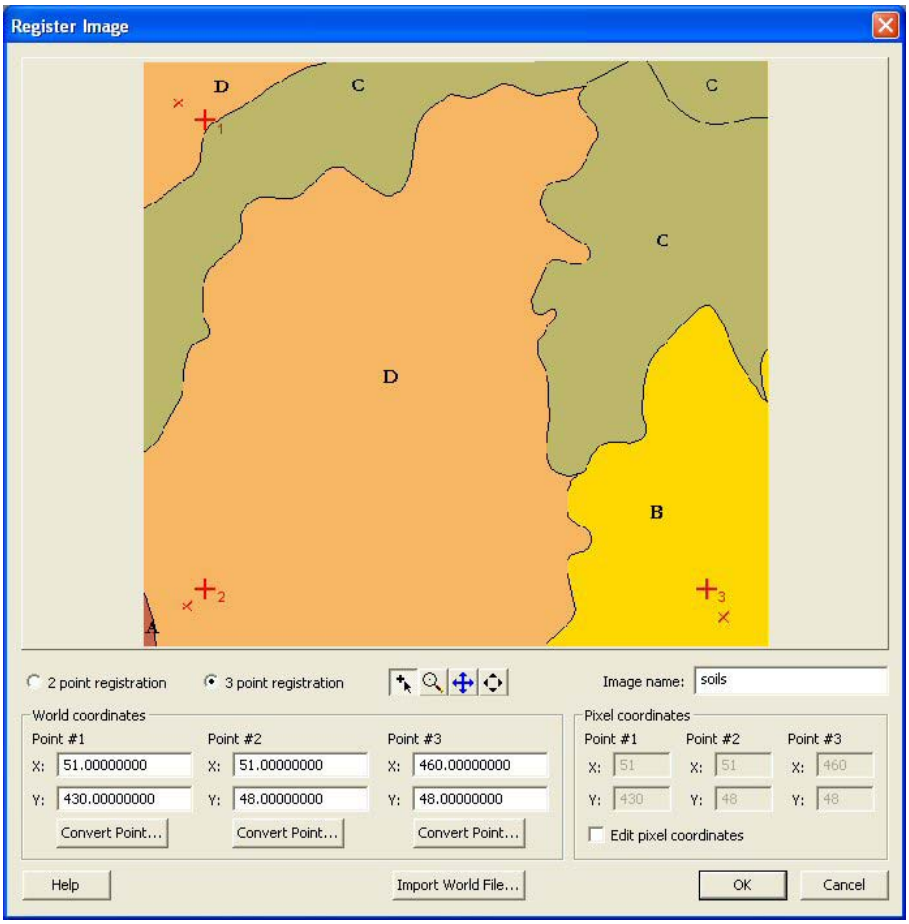


Figure 2-1: Moving + Marks in Registration Dialog

2.4.1 Conversion Calculator

We want to register images using the UTM x and y coordinates of each point, but we only know the latitude and longitude of the registration points. You can use the Convert Point… dialog to access the single point coordinate conversion dialog so that you can enter your coordinates in decimal degrees or degrees/minutes/seconds and convert to UTM (or any other) coordinates.

1. Choose the *Convert Point…* button from the *Point #1* area on the Register Image dialog.
2. Select *Geographic NAD 83(US)* from the *Convert From: Horizontal System:* drop down box.
3. Select *UTM NAD 83(US)* from the *Convert To: Horizontal System:* drop down box.
4. Set the Horizontal *Units* field to *Meters*
5. Select *UTM Zone 12 114W to 108W* from the *Convert To: UTM Zone* drop down box.
6. Set the *Vertical System* to *Local* and the *Units* field is *Meters*.
7. Find the latitude and longitude for Point 1 in Table 2-1.
8. Enter the appropriate *Degrees*, *Minutes*, and *Seconds* values for Point 1 in the edit boxes.
9. Choose the *Convert* button.
10. Select *OK* to exit the dialog.

The appropriate UTM x and y coordinates should appear in the X and Y edit boxes under Point 1 in the Register Image dialog.

1. Follow the same steps to convert the latitude and longitude coordinates of Points 2 and 3 to UTM coordinates (the settings in the conversion dialog should remain the same so that you only need to enter the coordinates for points 2 and 3).
2. Select *OK* on the *Register Image* dialog.

If the image appears distorted or crooked, you may have entered the coordinates incorrectly or placed the + marks inaccurately.

Table 2-1: Latitude and Longitude for Soils.tif

Point	Longitude	Latitude
1	112°28'55" W	38°41'06" N
2	112°28'38" W	38°34'36" N

3	112°19'49" W	38°34'34" N

1. Select ***File*** | ***New***

2. Select *NO* when asked if you want to save your changes

2.5 Aerial Photographs

It is often useful to use aerial photographs as background images in WMS. http://terraserver.microsoft.com has a large selection of satellite photos, aerial photographs, and images for locations throughout the United States. The aerial photographs from the TerraServer come with world files, simplifying the image registration process.

2.5.1 Download an Aerial Photograph from the Terraserver

To demonstrate TerraServer, we will download an aerial photograph of part of the Clear Creek watershed in Richfield, Utah.

If you are unable to connect to the internet at this point, or if you have difficulties obtaining the file(s) you can skip ahead to section 2.5.2 below and use the ClearCreek.jpg and ClearCreek.jgw files previously downloaded.

1. Go to http://terraserver.microsoft.com.

2. Type Joseph, UT in the Search TerraServer edit box shown in Figure 2-2.

Figure 2-2: Search TerraServer Edit Box and GO button

1. Choose *GO* (see Figure 2-2).

2. Choose the *Topo Map 7/1/1980* for *Joseph, Utah, United States* as shown in Figure 2-3.

Even though we are searching for an aerial photo, we will look at the topographic map, as it is easier to find specific features on.

Place Search
Find Results for 'Joseph, UT'
Place Name
Available Image
1 Joseph, Utah, United States
Aerial Photo 8/17/1993
Topo Map 7/1/1980
New Place Search
Enter a name of city or popular place. You can enter a partial name, e.g. "San" to find all places beginning with "San". Click on the State / Country drop-down list to narrow your search criteria to a specific state.
Place:
State:
Country:

Figure 2-3: Topo Map Image Link

1. Click on the *64 meter* zoom bar, as shown in Figure 2-4.

Figure 2-4: 64-meter Zoom Bar

1. Click on the location as indicated by the *X* in the lower left of Figure 2-5 to zoom in.

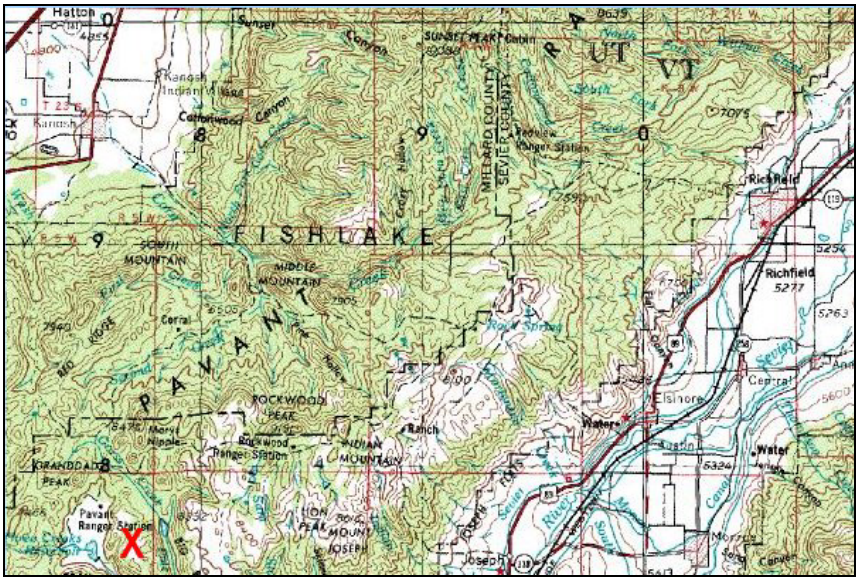


Figure 2-5: Zoom Location

1. Click on the *South* arrow, as shown in Figure 2-6, to pan to the South.

Figure 2-6: Pan South Arrow

1. Switch to the *Aerial Photo* view tab as shown in Figure 2-7.

Figure 2-7: Aerial Photo Tab

1. Choose the *Download* link as shown in Figure 2-7.

You will be taken to a screen with your aerial photo and instructions for downloading both the image and the world file. Follow the instructions to save your image and the corresponding GIS World Coordinates file. To save the image file you will right-click on the image and save it. To save the world file

follow the link on the lower-left side of the page. IMPORTANT: The world file must be saved as a text file and not an html file (the default). Be sure to change the file type to a text file when saving the world file.

Note that you should assign the image a name with a .jpg extension and the world file the same name, but with a *.jgw extension. If your browser appends any other extension to the file names, be sure to change them (it might put a .txt no matter what on the world file).

2.5.2 Import the Aerial Photograph Image

1. Switch to the *Map* module , if needed
2. Select ***Images* | *Import…***
3. Select *Jpeg image file (*.jpg, *.jpeg)* from the *Files of type* drop-down box
4. Open your downloaded file, or if you were unsuccessful then open "*ClearCreek.jpg*"

If you successfully downloaded the world file and it is named ClearCreek.jgw (note that when saving it may put a .txt extension on the file anyway) then it will open in WMS and be registered. If your world file has some other name, or does not end with .jgw then you can import it using the following three commands (skip these commands if it is already registered in WMS)

5. Choose the *Import World File* button on the Register Image dialog.
6. Open your downloaded world file (you might have to look for files with a .txt extension).
7. Select *OK* on the Register Image dialog.

You have successfully opened an aerial photograph and registered it in WMS.

2.6 Conclusion

In this tutorial, you were taught how to import several types of images into WMS. You learned how to georeference images and how to save the georeferenced images. In particular, you should know:

1. How to use geotiff files
2. What resampling an image means
3. How to use world files.

4. How to register scanned images
5. What a WMS image file is and how to save one
6. How to use aerial photographs

CHAPTER 3

Basic Feature Objects

Feature objects are points, lines, and polygons organized in coverages by different attribute sets such as drainage features, land use, soils, time travel paths, cross sections, etc. A synonymous word for coverage would be a layer (AutoCAD), theme (Arc View), or level (Micro Station). Simply put, a coverage contains points, lines, or polygons that have the same attribute set. For example, you don't want to store land use polygons in the same coverage or layer as drainage basin or soil polygons.

The primary coverage in WMS is the drainage coverage, which holds drainage boundary polygons, stream lines, and outlet nodes. Most of the other coverages are secondary to the drainage coverage and are used to "map" other hydrologic parameters such as travel time or curve numbers. Feature objects are equivalent to GIS vector data and so importing from GIS databases is one important way to create coverages in WMS. Another important method for creating feature object coverages is to digitize directly from the screen, using a georeferenced image in the background as a guide.

3.1 Objectives

In this workshop you will learn the basics for creating and importing feature objects and managing different coverages. This includes the following:

1. Creating and editing feature objects
2. Feature object attributes

3. Creating coverages and specifying attribute sets
4. Basic importing of shapefiles
5. Using images to create feature objects
6. Managing multiple coverages

3.2 Creating and Editing Feature Objects

The *Terrain Data, Drainage, and Map* modules are where the feature objects are created and manipulated. All feature objects are made from a set of points and the lines connecting the points. There are three main types of feature objects: points, arcs, and polygons. The following steps will teach you how to create and edit the different types of feature objects.

1. Switch to the *Map* module
2. Select ***File*** | ***Open…***
3. Open "*FeatureObjects.img*"

3.2.1 Creating Feature Arcs

1. Find the portion of the image labeled *Vertices, Nodes, and Arcs*.
2. Choose the *Create Feature Arc* tool
3. Single-click on the point labeled *1*
4. Double-click on point *2* to end the arc

While you are creating a feature arc, you can hit *Esc* to cancel, *Backspace* to back up one vertex, *Enter* or single-click to make a vertex, and double-click to end the arc. When WMS creates an arc, each end is a node and all points in the middle are vertices.

1. Single-click at point *3*, directly on top of the arc you just made

Notice how WMS automatically links the new arc to the existing arc and creates a node at the point of intersection.

1. Double-click at point *4* to end the arc
2. Single-click at point *5*

3. Double-click at point *6*

3.2.2 Snapping Arcs

Oftentimes you will have two arcs very close to each other that should share a common node, but do not. WMS has an option to snap these nodes together.

1. Find the portion of the image labeled *Vertices, Nodes, and Arcs*.
2. Choose the *Select Feature Point/Node* tool
3. Click on the node labeled *5*
4. Select ***Feature Objects* | *Clean…***
5. Make sure the *Snap selected nodes* option is checked
6. Select *OK*

If you look down at the bottom of the WMS screen, you will notice the help window is prompting you to select a snapping point.

7. Click on the main arc near the point labeled *7*

WMS will not snap the arcs together because no vertex exists at the point you selected. To be able to snap the two arcs together, you must insert a vertex at this point.

3.2.3 Inserting Vertices

1. Choose the *Create Feature Vertex* tool
2. Single-click on the arc where it is labeled *7*

A vertex is inserted here just as if you had clicked here when creating the arc originally. Now you can snap the two arcs together.

3. Choose the *Select Feature Point/Node* tool
4. Click on the node labeled *5*
5. Select **Feature Objects | Clean**.
6. Make sure the *Snap selected nodes* option is checked
7. Select *OK*

8. Select the vertex you just created.

WMS snaps the two arcs together and changes the vertex at point *7* to a node.

3.2.4 Deleting a Portion of an Arc

Now that the main arc you created has two nodes along its length, you can delete the center portion only.

1. Choose the *Select Feature Arc* tool
2. Select the portion of the arc between nodes *3* and *7*
3. Select the *Delete* key to delete the arc
4. Choose the *Create Feature Arc* tool
5. Click on the node labeled *3*
6. Double-click on the node labeled *7* to re-form the arc

3.2.5 Converting Vertices to Nodes

WMS uses vertices and nodes for different purposes. Sometimes you will need to change a vertex to a node or a node to a vertex.

1. Choose the *Create Feature Vertex* tool
2. Click on the arc somewhere between nodes *3* and *7*
3. Choose the *Select Feature Vertex* tool
4. Select the vertex you just made
5. Select ***Feature Objects | Vertex <-> Node***

You should now see a red node at this location.

3.2.6 Converting Nodes to Vertices

Just as you can change vertices to nodes, you can change nodes to vertices.

1. Choose the *Select Feature Point/Node* tool
2. Click on the node you just converted

3. Select **Feature Objects | Vertex <-> Node**

You can see that the node has been changed back to a vertex.

3.2.7 Building Polygons

Find the portion of the image labeled *Polygons*.

1. Choose the *Create Feature Arc* tool
2. Single-click at the point labeled *1* on polygon *A*
3. Single-click on points *2* through *10*
4. Single-click on point *1* again to end.
5. Trace polygon *B* in the same manner

You should now have two closed loops made out of the arcs just created. They are not polygons at this time – they are still just arcs.

6. Select ***Feature Objects | Build Polygon***
7. Select *OK* to use all the arcs

The two polygons should now be drawn with a thick line instead of the thinner arc lines. Polygons are built from their constituent arcs.

3.2.8 Assigning Attributes

Each of the nodes, arcs, and polygons you created were created with default properties or attributes. WMS allows you to change the attributes of feature objects.

1. While holding the *Shift* key (multi-select) down select all 5 arc sections in the *Vertices, Nodes, and Arcs* portion of the image
2. Select ***Feature Objects | Attributes…***

A dialog will come up allowing you to choose whether you want the arcs to have the *Generic*, *Stream*, *Lake*, or *Ridge* attribute.

3. Select the *Stream* option
4. Select *OK*

The arcs should now be colored blue. Each arc portion should have a small blue arrow on it. These arrows show the way the stream you have created

flows. The original direction you created the arc determines the way the stream flows now. Stream arcs should always be created from downstream to upstream. You should also be able to see that the lower node on the arc looks different now. WMS has automatically changed it to a drainage outlet instead of a generic node.

5. Choose the *Select Feature Point/Node* tool
6. Double click on the lower node (now an outlet)

A dialog comes up showing that the node now has the *Drainage outlet* attribute.

7. Select *OK*

Just as you can change the attributes of arcs and nodes, you can change the attributes of polygons.

8. Choose the *Select Feature Polygon* tool
9. Double-click anywhere inside Polygon *A* in the *Polygons* portion of the image
10. Select the *Drainage boundary* type
11. Select *OK*

Polygon A should now be drawn in a thick colored line instead of a black one.

12. Double-click anywhere inside Polygon *B*
13. Select the *Lake/Reservoir* type
14. Select *OK*

Polygon B should now be drawn in light blue.

3.3 Using Shape Files to Create Feature Objects

One of the most important features of WMS is the ability to automatically create feature objects using shapefiles. We will now import a streams shapefile and a basins shapefile and convert them to streams and basins.

1. Select ***File | New***
2. Select *NO* when asked if you want to save your changes

3. Switch to the *GIS* module

You will import shapefile data differently depending on whether the computer you are working on has ArcInfo installed on it or not. For this tutorial, the two ways are basically equivalent. However, if you have ArcInfo installed, you have access to more data types and display options. First, you will import a shapefile without ArcInfo.

3.3.1 Without ArcInfo

1. Select ***Data | Add Shapefile Data***.
2. Open "*streams.shp*"

In order for the shapefile to work correctly, streams.dbf and streams.shx must be located in the same directory as streams.shp. This is true for all shapefiles.

1. Choose the *Select Shapes* tool
2. Draw a box around all the shapes to select them all
3. Select ***Mapping | Shapes -> Feature Objects***

This is the GIS to Feature Objects Wizard. It is used to map shapefile data to feature objects in WMS.

1. Choose *Next*

The spreadsheet that is presented next shows each shapefile attribute in bold letters. In this file, you should see DRAINTYPE, LENGTH, SLOPE, etc. in bold letters. Underneath each attribute is a dropdown box containing the WMS attributes you can choose to map the shapefile attributes to.

1. Select *Drainage Arc type* from the dropdown box below *DRAINTYPE*
2. Select *Stream length* from the dropdown box below *LENGTH*
3. Select *Stream slope* from the dropdown box below *SLOPE*
4. Select *Not mapped* from the dropdown box below *DMANNINGS*

This attribute cannot be mapped because there is not a corresponding WMS attribute available to map it to.

1. Select *Stream basin id* from the dropdown box below *BASINID*

You can scroll through the mapping spreadsheet to see the value that is assigned to each attribute for each shape.

1. Choose *Next*

2. Choose *Finish*

You have now imported a shapefile containing streams and basins, converted all the shapes to WMS feature objects, and mapped data from the original shapefile to WMS attributes.

3.3.2 With ArcView

If you have ArcView (or ArcGIS) installed on your computer, you can choose to import shapefiles using ArcObjects capabilities. If you do not have ArcView then skip to the next section.

1. Select ***File | New***

2. Select *NO* when asked if you want to save your changes

3. Select ***Data | Enable Arc Objects***

If the computer you are working on does not have ArcInfo installed, the Data menu will not change. If ArcInfo is installed, the previously dimmed menu items should become undimmed.

1. Select ***Data | Add Data***

2. If ***Data | Add Data*** is dimmed, skip to section 3.4

3. Open "*streams.shp*"

In order for the shapefile to work correctly, streams.dbf and streams.shx must be located in the same directory as streams.shp. This is true for all shapefiles.

1. Choose the *Select Features* tool

2. Draw a box around all the shapes

3. Select ***Mapping | ArcObjects->Feature Objects***

4. This is the same GIS to Feature Objects Wizard you used in section 3.3.1. When you are finished mapping shapefile data to WMS attributes, choose *Next*, then *Finish*.

3.4 Creating Feature Objects Using Background Images

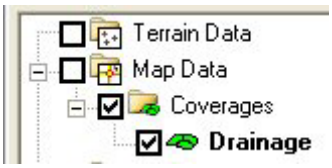
Another important feature of WMS is the ability to create feature objects using background images as guides. For instance, you may have a soil use map you want to read into WMS. The following procedure explains how this is done.

3.4.1 The Data Tree Window

First, you will need to create a new coverage by utilizing the Data Tree window on the right hand side of the WMS main window.

The Data Tree displays and allows users to manage the current coverages and data in WMS.

1. Select ***File*** | ***New***
2. Select *NO* when asked if you want to save your changes
3. Switch to the *Map* module , if necessary



In the data tree window, you should see a folder entitled *Map Data*. You should see the default coverage listed (always a Drainage coverage when beginning a new project) in the *Coverages* folder. From the Data Tree, you can manage the default coverage, make new coverages, delete coverages, edit coverage properties, and change the active coverage.

1. Right-click on the one existing coverage in the Data Tree
2. Select *Rename*
3. Enter **PracticeDrainage** for the new coverage name
4. Right-click on the *Coverages* folder
5. Select *New Coverage…*
6. From the *Coverage type* dropdown box, select *Soil Type*
7. Notice that the Coverage name is automatically changed to Soil Type
8. Select *OK*
9. Click on the *PracticeDrainage* coverage

You can see that this coverage shows up in color and bold, while the Soil Type drainage is in gray and regular font. This means that the PracticeDrainage coverage is the active coverage.

1. Select the *Soil Type* coverage to make it active
2. Uncheck the box next to the *PracticeDrainage* coverage

Now the PracticeDrainage coverage is not visible. Turn this coverage back on to make it visible again.

3.4.2 Reading in Images

Now that you have added a soil type coverage, you can read in the soils image.

1. Select ***Images*** | ***Import…***
2. Open "*soils.img"*

3.4.3 Manually Digitizing Feature Objects

1. Choose the *Create Feature Arc* tool
2. Starting anywhere on the border of the large orange area, outline the entire region labeled *D* with an arc

You can be as accurate as you like. If you wish, you can even zoom in to get a closer view of the image. You will have to end the arc by double clicking in order to be able to select any other tools, such as zoom, pan, or show all. Once you have zoomed to the location you want, you can just pick up where you left off. Remember that when you click near an existing vertex or arc, WMS will automatically snap the new arc to the existing one.

1. Outline all the other soil type polygons similarly
2. Select ***Feature Objects*** | ***Build Polygon***
3. Select *OK* to use all the arcs

Check to make sure that each soil use polygon is completely outlined. If one or more polygons do not build correctly, check to be sure that the arcs surrounding the polygons are completely closed.

3.4.4 Assigning Feature Polygon Attributes

Now that you have created the soil use polygons, you will need to assign the soil use attributes to the correct polygons.

1. Choose the *Select Feature Polygon* tool
2. Double click on the yellow polygon labeled *B*

The soil type mapping dialog should come up automatically. Just as in the first part of the tutorial, double-clicking on a feature object brings up a dialog allowing you to select or edit its attributes. Since you are now using a Soil Type coverage, the automatic attribute for a polygon is Soil Type.

1. Choose the *Add soil ID to list* button

2. Choose this button two more times, so that there are four soil types in the list box
3. Select *Soil ID 0* from the list
4. In the *selected soil properties* box, click on the *Soil Type* line
5. From the *Soil SCS ID* drop down box, select *Type A*
6. Select *Soil ID 1* from the list
7. Click on the *Soil Type* line, and select *Type B* from the drop down box
8. Assign *Type C* to *Soil ID 2* and *Type D* to *Soil ID 3*
9. Since the polygon we clicked on originally is Type B, select *Soil ID 1* from the list and choose the *Apply* button
10. Double click on the polygon labeled *A*
11. Select *Soil ID 0* from the list and choose *Apply*
12. Assign *Soil ID 2* to all the polygons labeled *C* and *Soil ID 3* to all the polygons labeled *D*
13. Make sure each polygon has the proper Soil ID assigned by double clicking on each and checking the soil type in the *Selected Soil Properties* box

3.5 More Feature Objects from Images

WMS handles land use images the same as it does soil type images. You will now read in a land use image, digitize polygons, and assign land uses. You need the land use image (and the resulting land use polygons) to be on its own coverage, so you will need to create a land use coverage.

1. Right-click on the *Coverages* folder in the Data Tree
2. Select *New Coverage…*
3. From the *Coverage type* dropdown box, select *Land Use*
4. Select *OK*

Usually you would open the land use image and digitize the land use polygons yourself. Here, you will open a completed file. To use the completed file, complete the following steps:

1. Select ***File*** | ***Open…***
2. Open "*luse.map*"

If you were digitizing the image yourself, you would complete the following steps:

1. Select ***Images*** | ***Import***…
2. Open "*luse.img*"

As outlined in Parts 3.4.3 and 3.4.4, you'd trace all the land use polygons with arc segments (while in the Map module), build polygons, and then map the land use ID's to the correct polygons.

3.6 Display Options

WMS has many display options to help you tailor the look of your project to your needs. You can change options such as polygon colors, presence of nodes and vertices, and legends using the Display Options command.

1. Activate the *Soil Type* coverage by single-clicking on its name in the Data Tree
2. Select ***Display*** | ***Display Options…***
3. On the *Map* tab, check the *Color Fill Polygons* box
4. Uncheck the *Points/Nodes* and *Vertices* boxes
5. Choose the *Soil Type Display Options* button
6. Select the first Soil ID listed in the list box and click on the color square to the right
7. Choose a new color from the color palette
8. Change the colors of the other soil groups uses if you desire
9. Select *OK*
10. Choose the General tab
11. Check the *Soil Type Legend* box
12. Select *OK* once again to exit the *Display Options* dialog.

You can continue to explore the display options if you wish. If you wanted to assign new colors to the land uses, you would need to make the land use coverage active before going back into the Display Options dialog.

3.6.1 Managing Coverages

Using the Data Tree window, you can choose to hide and/or show coverages and designate which coverage is the active coverage.

1. From the Data Tree window, toggle off the check boxes for the *Drainage* and the *Soil Type* coverages

2. Click on the *Land Use* coverage so it will be active

Now only the land use coverage will be visible on the screen. The other coverages still exist; they simply will not show on the screen until you turn their visibility back on.

3.7 Conclusion

In this workshop you should have learned the basics for creating and importing feature objects and managing different coverages. Both these concepts are central to your understanding of WMS. You should now be able to:

1. Create and edit feature objects

2. Set feature object attributes

3. Create coverages and specify coverage attribute sets

4. Import shapefiles

5. Use images to create feature objects

6. Manage multiple coverages

7. Import and edit feature objects from DXF data

CHAPTER 4

DEM Basics

Digital Elevation Models (DEMs) are the most commonly available digital elevation source and therefore an important part of using WMS for watershed characterization. A DEM is a rigid data structure that contains a two-dimensional array of elevations where the spacing between elevations is constant in the x and y directions. In the US DEMs are downloadable from the internet at 30-meter (1:24,000 map series) and 90-meter (1:250000 map series). The USGS has recently deployed the National Elevation Dataset which is a continuous elevation map at 30-meter resolution. Blocks of 100 MB or less can be downloaded for free from the NED website.

The Arc/Info ASCII grid format is common throughout the GIS world and is common outside the US. The basics of downloading, importing, editing, and displaying DEMs will be demonstrated in this workshop. Actually using the DEM for watershed delineation is the subject of the next workshop.

4.1 Objectives

In this workshop you will learn the basics of importing, viewing and preparing DEMs for automated watershed delineation. This includes the following:

1. Importing USGS DEMs from different formats
2. Tiling multiple DEMs together
3. Editing DEM elevations

4. Setting DEM display options

4.2 Getting DEMs from the Internet

In this part of the tutorial, you will learn how to download DEM data from WebMET, The Meteorological Resource Center, at http://www.webgis.com, from GeoCommunity/GIS Data Depot at http://www.gisdatadepot.com or http://www.geocomm.com, and from the National Elevation Data website at http://gisdata.usgs.net/ned/. Instead of book-marking all these sites, you can bookmark http://www.emrl.byu.edu/gsda. This site contains links to many sites where you can get DEM data.

If you do not have an internet connection you can still work through this tutorial, using the files which have been downloaded already and placed in the tutorials directories, or you may want to skip ahead to section 4.3 now.

4.2.1 WebMET

WebMET provides old style DEMs – 7.5 minute USGS quad DEMs.

1. Go to www.webgis.com
2. Select the *United States (7.5m)* link in the *Terrain Data* area
3. Click on *Utah* (UT)
4. Choose the *List Counties Alphabetically* link found underneath the map
5. Select *Sevier* County
6. Scroll down until you see *Trail Mountain*
7. Select the *Trail Mountain* link
8. Your browser should allow you to save the file to your computer. Choose a spot to save it in
9. Unzip the file using WinZip or another unzipping utility. The unzipped file should have a *.dem* extension

You are now able to use this DEM directly in WMS. If you are not connected to the internet or you were unsuccessful downloading the Trail Mountain DEM then you can use the file (trailmountain.dem) included with the tutorial data.

1. In WMS, switch to the *Terrain Data* module

2. Select ***File*** | ***Open…***

3. Open the **.dem* file you just unzipped

4. Select *OK*

WMS should read in the DEM, and it should look similar to other .dem files you have read in.

4.2.2 GeoCommunity/GIS Data Depot

The GeoCommunity/GIS Data Depot website provides the new style SDTS DEMs. If you want to learn more about the SDTS style of DEMs, the GIS Data Depot has an article you can read at http://data.geocomm.com/sdts/ (Aug 8, 2002). You will be asked to create a GeoCommunity account in order to download data from this website.

1. Go to www.gisdatadepot.com

2. From the list of topics on the left column, click on *USGS DEMs* or scroll down until you see the heading *USGS DEMs* on the main page

3. Scroll down to *DOWNLOAD DEM DATA HERE* and click on the link

4. Scroll down and select *Utah*

5. Select *Sevier* County

6. Select *Digital Elevation Models (DEM) – 24K*

7. Scroll down to *Trail Mountain, UT (38112e4)* and click on the green download button

8. Log in or create a GeoCommunity account

9. Select the *1666653.DEM.SDTS.TAR.GZ (30 meter)* link.

10. Save the file on your computer. If you are unsuccessful doing this you will find the downloaded file included and unzipped with the WMS tutorial data. It has been unzipped into the trailmtnsdtsdem directory.

11. Use an unzipping utility such as Winzip to unzip the downloaded file.

If you wish, you can try to open up this DEM in WMS.

1. Select ***File*** | ***New***

2. Switch to the *Terrain Data* module

3. Select ***File*** | ***Open…***

4. Open any one of the *.ddf* files you just unzipped. It does not matter which of the several .ddf files you pick, WMS will automatically determine which files contain the necessary data.

5. Select *OK*

WMS should read in the DEM, and it should look similar to other .dem files you have read in. If you are interested in experimenting with a DEM that has a higher resolution you can load the 10-meter DEM for the Trail Mountain quadrangle. You will find the downloaded (or you can download for yourself) 10 meter SDTS DEM in the 1699603.DEM.SDTS.TAR.GZ file.

4.2.3 USGS National Elevation Dataset

The USGS National Elevation Dataset (NED) provides continuous 1:24,000-scaled DEM data for all of the contiguous US and 1:63,360-scale DEM data for Alaska. To access this data, complete the following steps:

1. Go to http://seamless.usgs.gov, then click on *View and Download United States Data*. (You could also go to http://www.emrl.byu.edu/gsda and click on *DEM*, then click on *OBTAIN DEM DATA*, then click on the *CLICK HERE* to obtain 1:24,000 (30m) DEM data from the National Elevation Database.)

2. Choose the *Find* tool (the binoculars in the query toolbar) and type in **Richfield**

3. Click the *OK* button

4. Find the Richfield with a longitude of *112 04 59,99W* and a latitude of 38 46 21.00N, and click on *Go To*

5. Select the *Display Tab* on the right of the screen. Click on the arrow next to *Hydrography*.

6. Select the check boxes next to *National Atlas and NHD Streams* and *National Atlas and NHD Waterbodies* as shown in Figure 4-1. The map should automatically update.

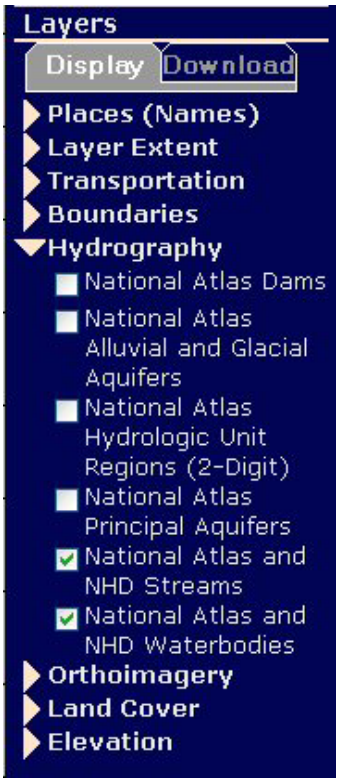


Figure 4-1: Display Layers

7. Use the *Zoom* tool to zoom in around the area shown in Figure 4-2.

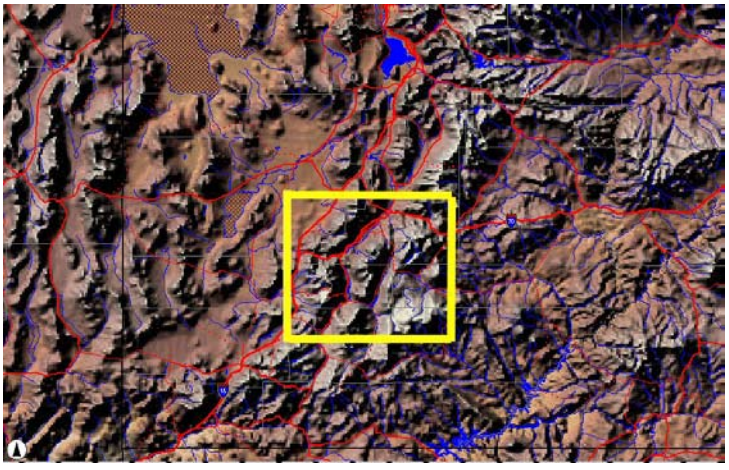


Figure 4-2: Zoom Window

1. Under the *Download* section, choose the *Download Area* tool

2. Make the rectangle shown in Figure 4-3. An order confirmation page should come up.

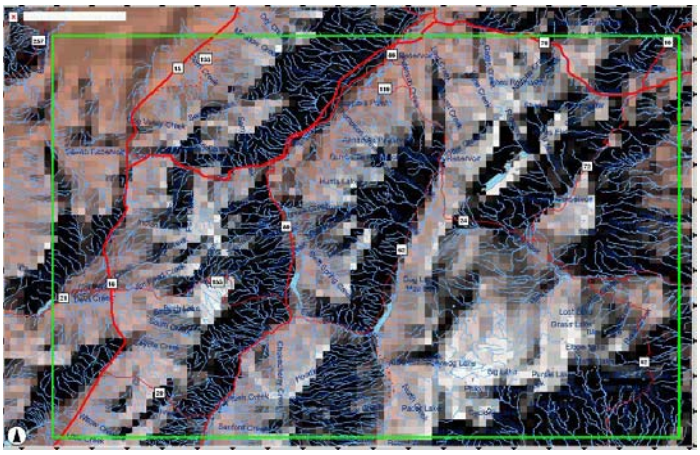


Figure 4-3: Download Selection Window

1. Select the *Modify Data Request* button.
2. Select *GridFloat* from the *Data Format* drop down box next to National Elevation Dataset (NED) 1/3 Arc Second
3. Select the *Save Changes & Return to Summary* button.
4. Select the *Download* link when the request summary page displays again. Save the .zip file to a location where you can unzip it.
5. Use WinZip, or any unzipping utility to unzip the file that you download. If you were not successful downloading the DEM data you can use the richfieldNED.zip file found with the tutorial data, which has been unzipped into the NED directory.

If you wish, you can try to open up this DEM in WMS.

1. Select ***File*** | ***New***
2. Switch to the *Terrain Data* module
3. Select ***File*** | ***Open…***
4. Open "*17133307.hdr"* (this is one of the files you just unzipped, or you can open it from the NED subdirectory).
5. Select *OK*

6. Select *No* when prompted to convert coordinates

WMS should read in the DEM, and it should be similar to other DEM files you have read in.

4.2.4 Arc/Info ASCII Grid Format DEMs

Many agencies manage their data with Arc/Info software, so you will often need GIS data in the ASCII grid format so that it can be used in WMS.

1. Select ***File*** | ***New***
2. Select ***File*** | ***Open…***
3. Open "*ArcInfoDEM.asc*"
4. Select *OK*

The Arc/Info grid is read in just as any other DEM.

4.3 Merging DEMs

The area you are studying may lie across two or more DEM quads. WMS is able to merge multiple DEM files that span quad sheets. To see how this works, complete the following steps:

1. Select ***File*** | ***New***
2. Select ***File*** | ***Open…***
3. Find and multi-select *josephpeak.dem, redridge.dem, marysvalecanyon.dem, and trailmountain.dem*.
4. Choose *Open*

You will be taken to the *Importing USGS DEMs* manager. The area covered by the DEM you have selected will be colored in a box near the center of the dialog. The boundaries of your DEM area will show up in the four edit boxes.

5. Select *OK*

By multi-selecting the DEMs you want, you can read in all the quads you need at the same time. WMS is able to read in an unlimited number of DEMs at a time (unless your computer runs out of memory). You can read in multiple DEMs in the standard USGS format (from the WebGIS site), the SDTS format,

or Arc/Info ASCII grid format, but you cannot mix and match formats in WMS (you could export any DEM in the Arc/Info ASCII grid format in order to get them all to a common format though). You cannot tile multiple DEMs from the NED site (this shouldn't be a problem though since the data from this site is "seamless" in the first place).

4.4 Trimming DEMs

WMS allows you to select the portion of the DEM you need and eliminate all of the surrounding elevation points. This can be done with an existing polygon you select, or you can enter the polygon interactively

4.4.1 Trimming DEMs

1. Switch to the *Terrain Data* module
2. Select ***DEM | Trim | Polygon…***
3. Choose the *Enter polygon interactively* option
4. Select *OK*
5. Click each of the four vertices of the polygon shown in Figure 4-4, double clicking on the last to close the polygon

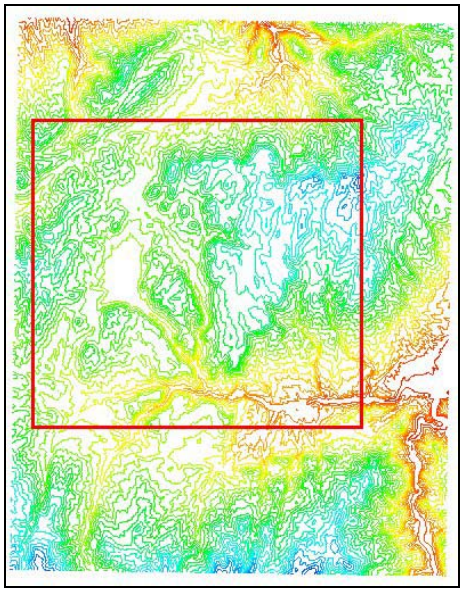


Figure 4-4: DEM Trim Area

4.5 Displaying DEMs

4.5.1 Contour Options

WMS has several options for displaying DEMs. First, you can change the contour display options by following these steps:

1. Select ***Display*** | ***Display Options…***
2. Select the *DEM* tab and choose the *Contours*... button. Make sure you are on the DEM tab, not the TIN tab
3. Under *Contour Interval*, change the number of contours in the edit box to **10**
4. Select *OK* twice

You can see that there are fewer contours and they are spread farther out now.

1. Select ***Display*** | ***Display Options…***
2. Select the *DEM* tab and choose the *Contours* button
3. Under *Contour Method*, select the *Color fill between contours* radio button

This option will 'color in' the contours.

1. Under *Bold and Label* check the *Display Legend* checkbox
2. Select *OK* twice

You can go back into the Contour options dialog and explore more of the contour display options if you wish.

4.5.2 Point Display Step

Next you will explore the point display step option:

1. Select ***Display*** | ***Display Options…***
2. Select the *DEM* tab
3. Change the number in the *Point display step* edit box to 4
4. Select *OK*

You can see that the display is not as smooth now.

1. Select ***Display | Display Options…***
2. Select the *DEM* tab
3. Change the *Point display step* to **12**
4. Select *OK*
5. Change the *Point display step* back to **2**

Raising the point display step will allow your DEM display to refresh faster, especially if your computer is slow. Although WMS is not drawing every DEM point, each point that was read in is still there, so changing the point display step does not change the accuracy.

4.5.3 Shading Options

Now you will explore the shading options:

1. Select ***Display | Shading Options…***
2. Toggle on *Use light source*
3. Click on the globe to move the light source and change the amount of ambient light
4. Select *OK*

Changing the position of the light source and the ambient light alters the display of the DEM. You can play around with these options to accentuate the elevations that are most important for you to see.

4.5.4 Views

Many times it is easier to see the DEM elevations from another view other than the straight overhead birds' eye view.

1. Select ***Display | View-Oblique View***. (Alternatively, you can also select the *Perspective View* macro)

It is now much easier to see the changes in elevation on the DEM. If the elevation relief cannot be seen clearly, you can change the Z-magnification to accentuate the elevation relief.

1. Select ***Display | View-Z Magnification***
2. Enter **3** in the edit box.

3. Select *OK*

The image will redraw on its' own. You will be able to see the elevation relief better now.

4.6 Conclusion

DEM data for the United States is found in several places on the internet. Data is available for most areas of the U.S. and some parts of other countries. DEM data is very useful for delineating watersheds in WMS.

In this workshop you should have learned how to do the following:

1. Import USGS DEMs from different formats
2. Tile multiple DEMs together and edit DEM elevations
3. Set DEM display options

CHAPTER 5

Using Triangulated Irregular Networks

Triangulated Irregular Networks (TINs) are constructed from a scattered set of x y z vertices. They can be used for visualization, as background elevations maps for generating new TINs or DEMs, or to perform basin delineation and drainage analysis. **WMS** has powerful tools for importing and manipulating this type of terrain data.

5.1 Objectives

The following topics will be covered in this tutorial:

1. Importing survey data
2. Digitizing data
3. Triangulation
4. Automated TIN editing
5. Manual TIN editing
6. Creating a TIN using a conceptual model

7. Converting a TIN to a DEM
8. Exporting data to CAD

5.2 Importing Survey Data

1. Switch to the Terrain Data module
2. Select ***File*** | ***Open…***
3. Open "*trailmountain.tif*"
4. Select ***File*** | ***Open…***
5. Open "*surveytm.txt*"
6. Leave the default settings. This is a tab delimited file exported from Excel. Select *Next*
7. For *WMS data type* choose **Survey Data**
8. In the *File preview* spreadsheet, map the first, second, and third column types to X, Y, and Z, respectively.
9. Select *Finish*
10. Select ***Display*** | ***Display Options…***
11. Toggle *Unlocked Vertices* on
12. Select *OK*
13. Zoom in around the vertices

5.3 Digitizing Data

1. Select ***TIN*** | ***TIN Options…***
2. Choose *Digitizing mode*
3. Select *OK*

4. Select the *Add Vertices* tool from the TIN toolbar (make sure you do not select the *Create Feature Point* tool from the Feature Objects toolbar—it looks the same as the *Add Vertices* tool).

5. Enter a Z value of **6800** ft in the edit window at the top of the screen

6. Digitize the 6800 ft contour by using the background image to add vertices

7. Select ***TIN*** | ***TIN Options…***

8. Choose *Non-digitizing mode*

9. Select *OK*

5.4 Triangulation

In order to edit the TIN according to the steps in this tutorial we will delete the existing data and read in a TIN file before we triangulate the data.

1. In the *Terrain Data* folder in the Data Tree right-click on the *"surveytm"* TIN and select ***Delete***

2. Select ***File*** | ***Open…***

3. Open *"digitizetm.tin"*

4. Zoom in around the vertices

5. Select ***TIN*** | ***Triangle*** | ***Triangulate***

5.5 Automated TIN Editing

WMS has automated methods of editing TINs so that they provide a representation of terrain that is useful for drainage analysis. These methods include selecting thin boundary triangles, transforming data, and eliminating flat triangles and pits. Thin boundary triangles, which result from triangulating the data, do not necessarily represent the terrain and are automatically deleted at the end of the triangulation process. Data transformations are especially useful for converting elevations between feet and meters. Flat triangles need to be eliminated for drainage analysis. Inaccurate basin delineation can result from pits because water flows into the pits rather than to the actual outlet.

5.5.1 Transformations

Run the cursor over the TIN and notice that the z values in the help strip at the bottom of the WMS window are in feet.

1. Select ***TIN*** | ***Vertex*** | ***Transform…***
2. Enter **0.3048** for the *Z Scale* value to scale the elevations in feet to meters
3. Toggle the *Frame image after transformation* option off
4. Select *OK*

Run the cursor over the TIN and notice that the z values in the help strip have changed to meters.

5.5.2 Flat Triangles

1. Select ***Display*** | ***Display Options…***
2. Toggle *Flat Triangles* on
3. Select *OK*
4. Select ***TIN*** | ***Triangle*** | ***Remove Flat Triangles***

5.5.3 Pits

1. Select ***Display*** | ***Display Options…***
2. Toggle *Pits* on
3. Select *OK*
4. Select ***TIN*** | ***Vertex*** | ***Smooth Pits***

Some pits may remain even after using this command.

5.5.4 Delineate Basin

1. Zoom in to the rectangle shown around Pit 1 in Figure 5-1

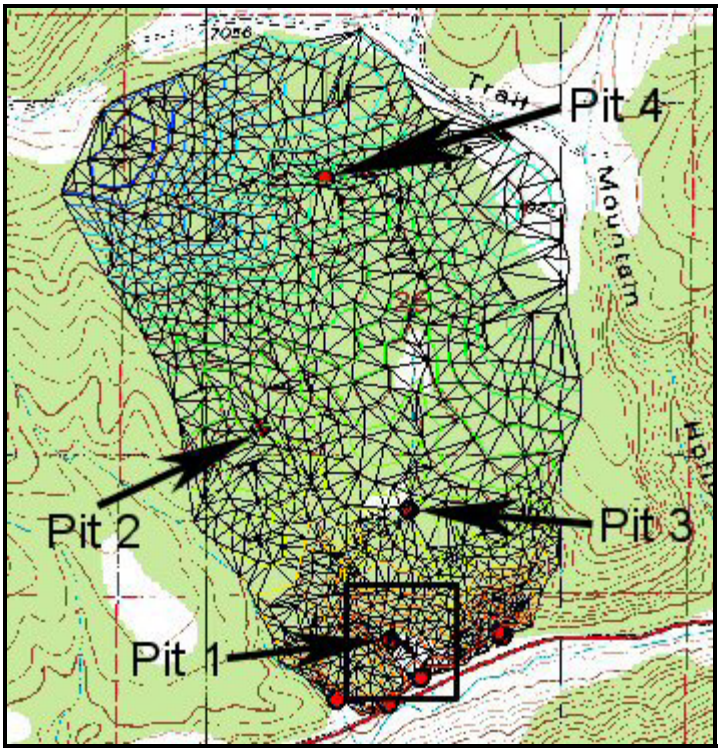


Figure 5-1: Remaining pits

2. *Select TIN Vertices* tool . Then, double-click on the TIN vertex shown in Figure 5-2 and change the type from *Generic* to *Drainage outlet*

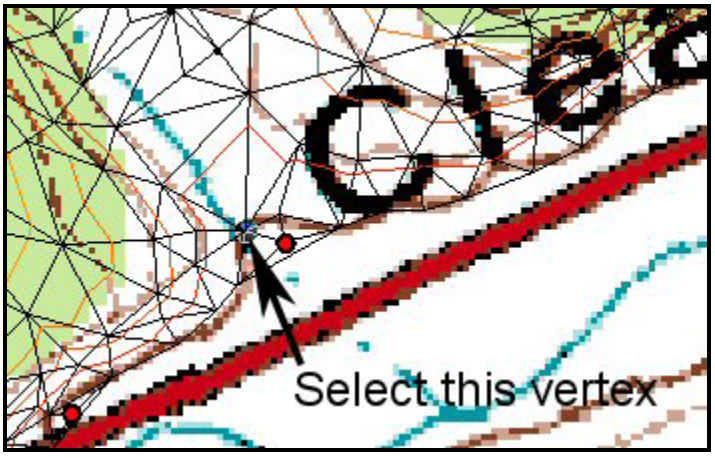


Figure 5-2: Outlet location

3. Select *OK*

4. Switch to the Drainage module

5. Select ***Display | Display Options…***

6. Toggle *Drainage Basin Boundaries* on and change the color to yellow so that they are more visible

7. Select *OK*

8. Select ***TIN | Define Basins***

9. Use the *Flow Path* tool to draw flow paths

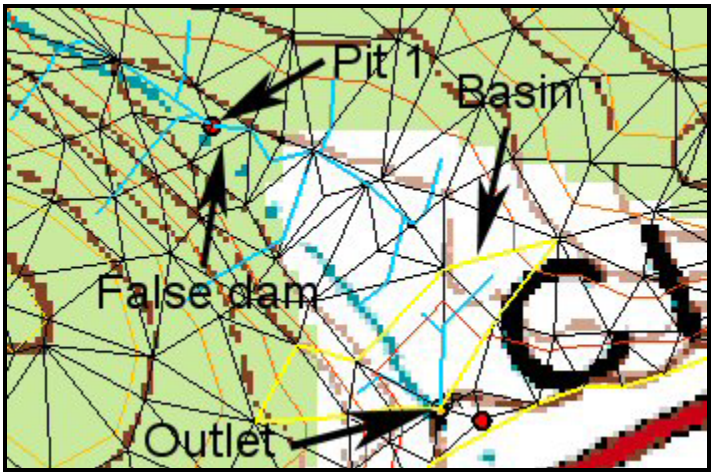


Figure 5-3: Flow paths

In Figure 5-3 notice that most flow paths flow into Pit 1. This does not allow the flow to make it to the designated outlet point and results in inaccurate basin delineation. Also notice the false dams that have been created from the triangulation. False dams generally occur where a triangle edge lies perpendicular to the stream

5.6 Manual TIN Editing

Manual methods of editing TINs include inserting breaklines, swapping triangle edges, inserting or deleting vertices, and editing vertex elevation and location. Breaklines are useful along streams and ridges by forcing triangle edges to lie along the breakline during triangulation. Swapping triangle edges can eliminate false dams and pits and smooth contours. Manipulating vertices also alters drainage characteristics of the TIN.

5.6.1 Pit 1 – Inserting a Breakline

1. Switch to the Terrain Data module

2. Select the *Create breakline* tool

3. Click on the outlet and then on an upstream vertex as shown in Figure 5-4 to create a breakline

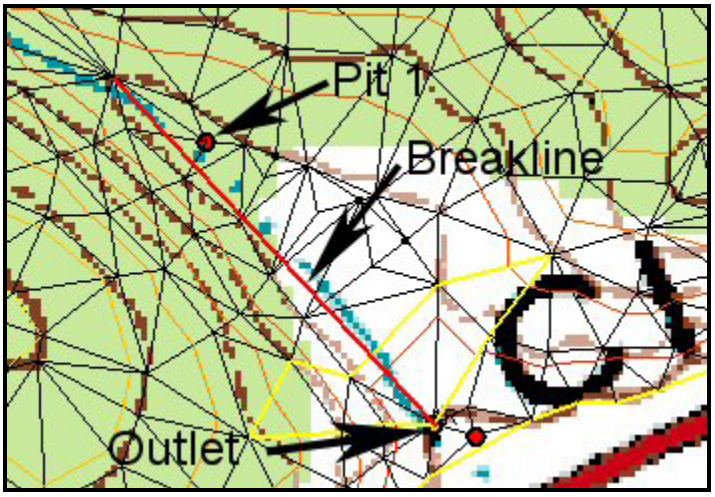


Figure 5-4: Breakline

4. Select ***TIN | Triangle | Insert Breakline(s)…***

Triangle edges are forced into the TIN and the false dams and pit are removed.

5.6.2 Pit 1 – Swapping Edges

1. Use the *Swap triangle edges* tool to swap edges by clicking on or near the triangle edge to be swapped for the three numbered triangles in Figure 5-5

Figure 5-5: Jagged contours

Figure 5-6 shows the smooth contours after swapping edges.

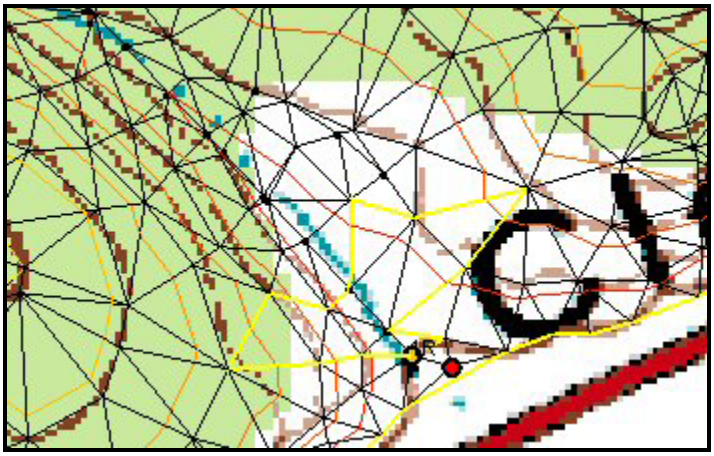


Figure 5-6: Contours after swapping edges

2. Zoom out so that you can see the entire TIN
3. Switch to the Drainage module
4. Select ***TIN*** | ***Define Basins***

5.6.3 Pit 2 – Inserting a Breakline

1. Zoom in on Pit 2 as shown in Figure 5-7

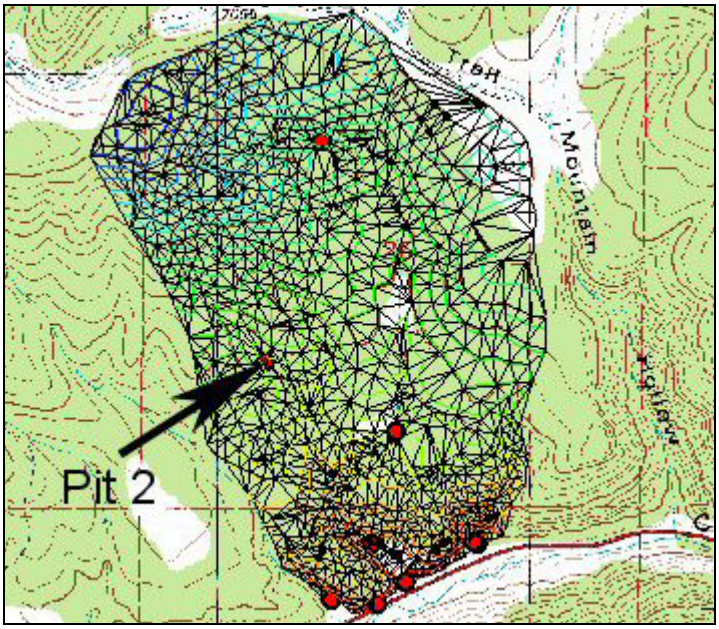


Figure 5-7: Pit 2

2. Switch to the Terrain Data module

5. Select the *Create breakline* tool

6. Create a breakline as shown in Figure 5-8

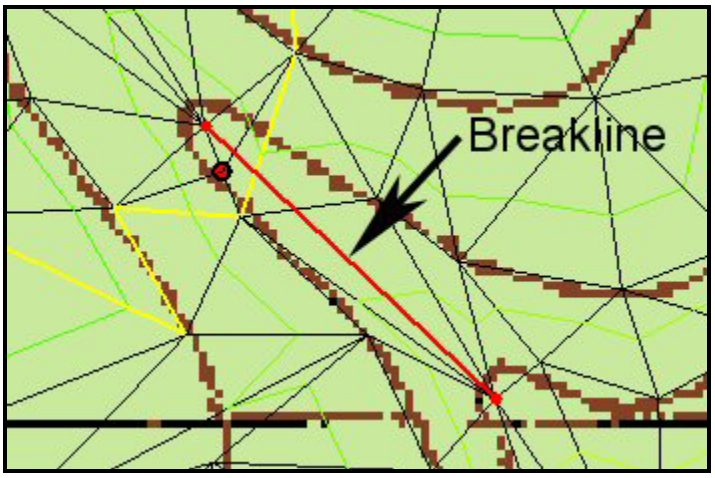


Figure 5-8: Pit 2 breakline

7. Select ***TIN | Triangle | Insert Breakline(s)…***

3. Zoom out so that you can see the entire TIN

4. Switch to the Drainage module

8. Select ***TIN | Define Basins***

5.6.4 Pit 3 – Swapping Edges

1. Zoom in on Pit 3 as shown in Figure 5-9

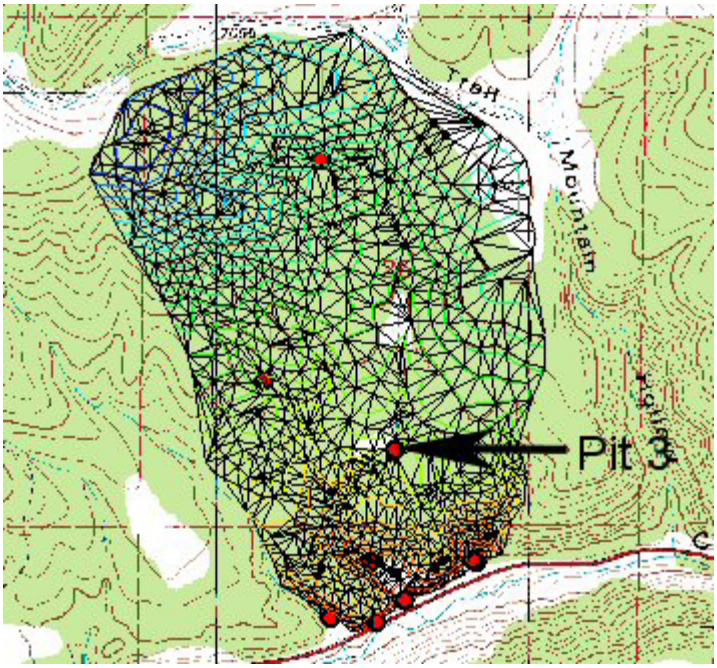


Figure 5-9: Pit 3

2. Use the *Swap triangle edges* tool to swap edges by clicking on or near the triangle edge to be swapped for the three numbered triangles in Figure 5-10 and Figure 5-11

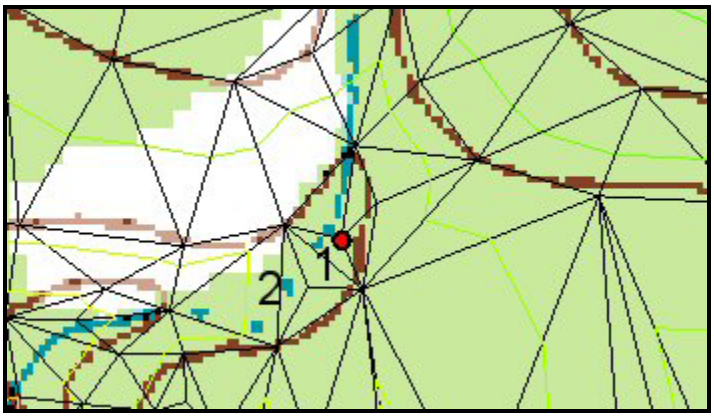


Figure 5-10: Pit 2 swap edges

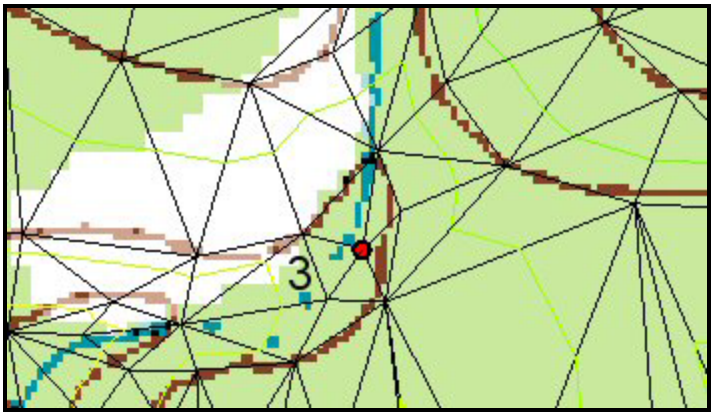


Figure 5-11: Pit 2 swap edges

3. Zoom out so that you can see the entire TIN

4. Select ***TIN | Define Basins***

5.6.5 Pit 4 – Swapping Edges

1. Zoom in on Pit 4 as shown in Figure 5-12

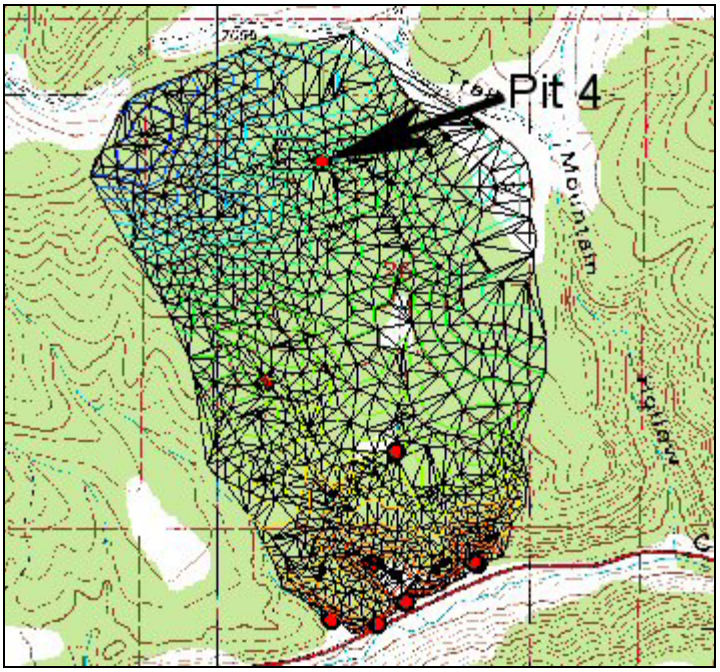


Figure 5-12: Pit 4

2. Switch to the Terrain Data module

3. Use the *Swap triangle edges* tool to swap edges by clicking on or near the triangle edge shown in Figure 5-13

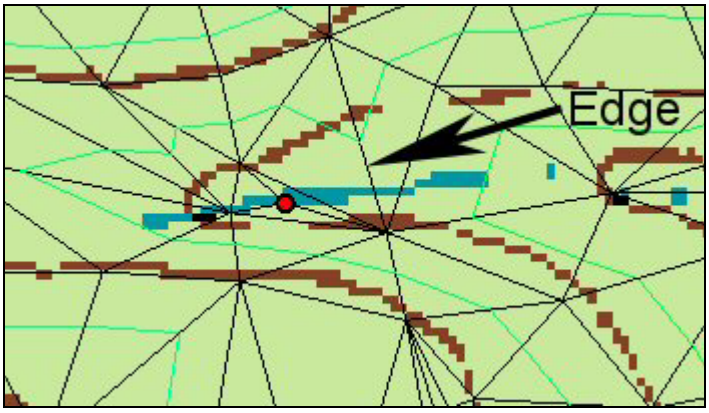


Figure 5-13: Pit 4 swap edge

When we swap edge 1, a flat triangle is created. Fixing the flat triangle will remove both the flat triangle and the pit.

4. Select ***TIN*** | ***Triangle*** | ***Remove Flat Triangles***

5.6.6 Delineate Basin

1. Zoom out so that you can see the entire TIN
2. Switch to the Drainage module
3. Select ***TIN*** | ***Define Basins***
4. Select ***TIN*** | ***Refine Boundaries***
5. Select ***TIN*** | ***Compute Basin Data…***
6. Select *OK*

5.7 Creating a TIN Using a Conceptual Model

A conceptual model built using feature object data can be used to create a TIN. This is important since an arbitrary triangulation of scattered data points may not produce a TIN that is suitable for drainage analysis (i.e. triangle edges enforced along streams and ridges). If you have a set of scattered data points it may be a good idea to triangulate them and then treat them as a background elevation map for creating a new TIN using a conceptual model.

5.7.1 Open Files

1. Select ***File*** | ***New***
2. Select *NO* when asked if you want to save changes
3. Select ***File*** | ***Open…***
4. Open "*trailmountain.tif*"
5. Select ***File*** | ***Open…***
6. Open *"digitizetm.tin"*
7. Zoom in around the x y z data

5.7.2 Triangulation

1. Switch to the Terrain Data module
2. Select ***Display*** | ***Display Options…***
3. Toggle *Flat Triangles* and *Pits* off
4. Select *OK*
5. Select ***TIN*** | ***Triangle*** | ***Triangulate***

5.7.3 Creating Feature Arcs for a Conceptual Model

1. Select ***Display*** | ***Display Options…***
2. On the TIN tab toggle *Unlocked Vertices* and *Triangles* off
3. On the TIN Drainage tab toggle *Drainage Basin Boundaries* off
4. Select *OK*
5. Switch to the Map module
6. Expand the *Map Data* folder in the Data Tree if necessary
7. Right-click on the *"General"* coverage and select ***Properties…***
8. Change the *Coverage Type* to **Drainage**

9. Select *OK*

10. Select the *Create Feature Arc* tool

11. Select ***Feature Objects*** | ***Attributes…***

12. Make sure the current feature arc type is set to *Generic* and select *OK*

13. Use the *Create Feature Arc* tool to create an arc that begins and ends at the outlet and follows along the inside edge of the TIN boundary as shown in Figure 5-14

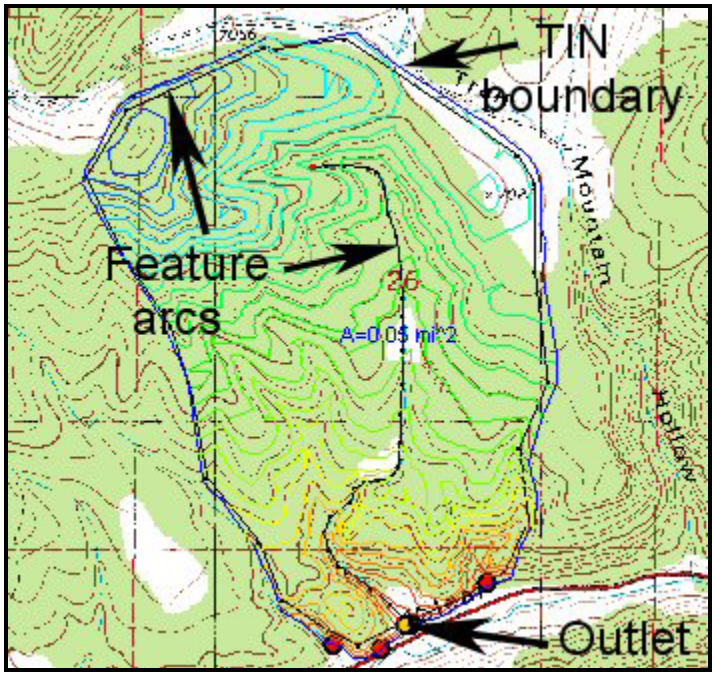


Figure 5-14: Creating an arc along the TIN boundary

14. Use the *Create Feature Arc* tool to create an arc starting at the outlet that follows the stream on the background image as shown in Figure 5-14. Make sure you create the arc from downstream to upstream.

15. Use the *Select Feature Arc* tool to select the arc along the stream

16. Select ***Feature Objects*** | ***Attributes…***

17. Choose a *Feature Arc Type* of **Stream**

18. Select *OK*

19. Use the *Select Feature Arc* tool to select the arc along the TIN boundary

20. Select ***Feature Objects*** | ***Build Polygon***

5.7.4 Redistribute Vertices

The size of the triangles created during triangulation depends on the spacing between vertices.

1. Select ***Edit*** | ***Select All*** to select all feature arcs
2. Select ***Feature Objects*** | ***Redistribute…***
3. Enter a *Spacing* of **60**
4. Select *OK*

5.7.5 Create TIN

1. Select ***Display*** | ***Display Options…***
2. On the TIN tab toggle *Triangles* on
3. Select *OK*
4. Select ***Feature Objects*** | ***Create TIN…***
5. Select *No*
6. Toggle the *Display triangulation process* option on
7. Select *OK*

The existing TIN is replaced with a new TIN that was created using the feature arcs that are part of the conceptual model.

5.7.6 Flat Triangles and Pits

1. Switch to the Terrain Data module
2. Select ***Display*** | ***Display Options…***
3. Toggle *Flat Triangles* and *Pits* on
4. Select *OK*

5. Select ***TIN*** | ***Vertex*** | ***Smooth Pits***

5.7.7 Basin Delineation

1. Switch to the Drainage module
2. Select ***TIN*** | ***Define Basins***
3. Select ***TIN*** | ***Refine Boundaries***
4. Select ***TIN*** | ***Compute Basin Data…***
5. Select *OK*

5.8 Convert to DEM

1. Switch to the Terrain Data module
2. Select ***TIN*** | ***Conversion*** | ***TIN->DEM…***
3. Enter a *cell width* and *cell height* of **10**
4. Select *OK*
5. Select *Yes*
6. Switch to the Map module
7. Expand the *Map Data* folder in the Data Tree and hide the display of *"trailmountain.TIF"* by toggling the visibility checkbox off
8. Switch to the Drainage module
9. Zoom in around the DEM contours
10. Select ***DEM*** | ***Compute TOPAZ Flow Data…***
11. Select *OK*
12. Select *OK*
13. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
14. Select ***Display*** | ***Display Options…***

15. On the DEM tab change the *Minimum Accumulation For Display* to 0.005 mi^2

16. Select *OK*

17. Select the *Create Outlet Point* tool

18. Create an outlet as shown in Figure 5-15

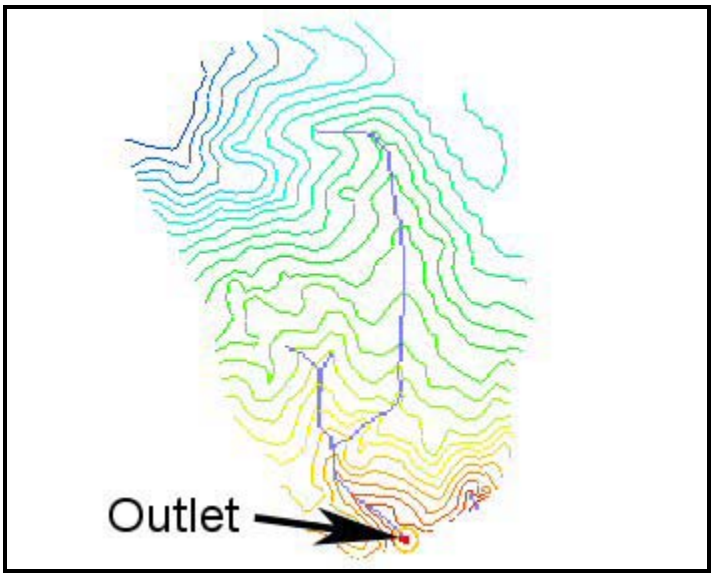


Figure 5-15: DEM outlet

19. Select ***DEM | Delineate Basins Wizard***

20. Select *OK*

21. Select *OK*

22. A new drainage boundary is created using the DEM data. The basin data for this drainage basin is computed.

5.9 Exporting Data to CAD

Data that is visible on the screen can be converted to CAD data and then saved for use in CAD programs.

1. Switch to the Map module

2. Select ***CAD | Data -> CAD***

In order to view only the newly created CAD data we will hide all other data.

3. Hide the *Map Data* folder in the Data Tree by toggling its visibility checkbox off
4. Switch to the Terrain Data module and hide the *Terrain Data* folder in the Data Tree
5. Select ***Display | Display Options…***
6. On the DEM tab toggle *Color Fill Drainage Basins* and *Fill Basin Boundary Only* off
7. Select *OK*
8. Switch to the Map module
9. Select ***CAD | Display Options…***
10. In the *Visibility* column toggle the *"Drainage_arcs"* off
11. Select *Apply*. Notice the CAD data that disappears when selecting this button.
12. Select *Cancel*
13. Select ***File | Save As…***
14. For *Save as type* choose the **DWG files (*.dwg)** filter
15. Select *Save*

CHAPTER 6

Advanced Feature Objects

In the previous workshop you learned how feature points, lines, and polygons are created and organized into coverages. In this workshop you will continue to learn about the creation and editing of feature objects, with a focus on creating drainage coverages, the primary coverage used by WMS to develop watershed models.

6.1 Objectives

In this workshop you will learn the basics for creating and importing feature objects and managing different coverages. This includes the following:

1. Using feature object drainage coverages for watershed delineation
2. Advanced feature object editing functions
3. Assigning appropriate feature object attributes
4. Importing and editing feature objects from DXF data

This tutorial demonstrates how feature objects alone can be used to define hydrologic models. Feature objects may be digitized on-screen using a registered image, or imported as a shape file from an Arc/Info or ArcView layer. Using the drainage coverage type, features can be converted to streams, outlets, and basins to represent the watershed being modeled.

6.2 Creating a Watershed from Scratch with Feature Objects

By using a combination of stream arcs, outlet nodes, and basin polygons, you can develop an entire watershed even without the use of a digital terrain model. The watershed can be to scale or a schematic. Of course, if it were not to scale, polygon areas and stream lengths would not be valid for your hydrologic model.

In this section of the tutorial you will create the Aspen Grove watershed from an image of a scanned paper map with clearly marked streams and basin boundaries.

1. Switch to the *Map* module
2. Select ***File*** | ***Open…***
3. Open "*aspentrc.img*"

You should see a portion of a USGS quad map with basin boundaries outlined in red and the stream network in black.

6.2.1 Creating Basin Boundaries

We will begin by creating the basin boundaries, but it does not matter whether the basins or streams are created first.

1. Choose the *Create Arcs* tool
2. Select ***Feature Objects*** | ***Attributes…***
3. Make sure that the arc type is *Generic* and Select *OK*
4. Beginning at the outlet point (lower right) trace out the entire watershed boundary. You do not need to follow every detail; take as much time as you want. End by *double-clicking* near the same point where you began.
5. Now create each of the other three sub-basin boundary arcs on the interior of the watershed. Begin by clicking on a point near the junction in the center of the watershed and ending by double-clicking near the intersection of the arc previously created for the exterior boundary.

6.2.2 Creating the Stream Network

The stream network is created in much the same way the basin boundaries were. The only thing to note is that in the upper basin the basin boundary

comes very close to the stream. You will need to zoom in on this region in order to avoid conflicts with the snapping tolerance.

1. Select ***Feature Objects*** | ***Attributes…***

2. Choose the *Stream* feature arc type

3. Select *OK*

4. Create the main channel from the outlet of the watershed to the outlet point for the two upper basins. Begin by clicking near enough to the boundary arc at the outlet so that it snaps to it and end by double-clicking on the basin junction point.

5. Create the two branches of the lower basin by clicking on a point near the stream arc just created and double-clicking at the most upstream point of the branches in the image.

NOTE: As you create new vertices on stream arcs you should always do so from downstream to upstream.

1. Choose the *Zoom* tool

2. Zoom in on the region shown in Figure 6-1

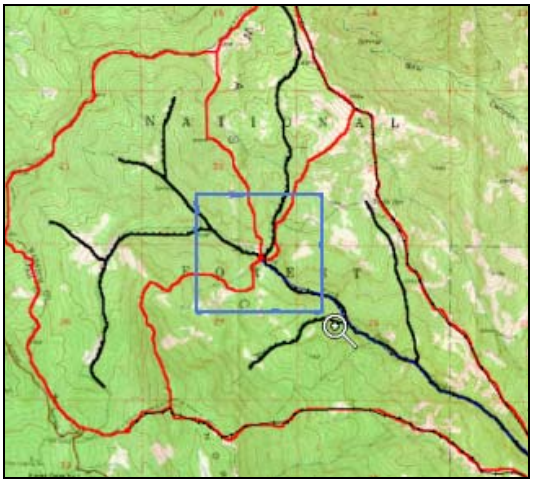


Figure 6-1: Junction of Main Channel in Aspen Grove Watershed.

1. Choose the *Create Arcs* tool

2. Create the initial portion of each portion of the stream by clicking on the junction point (intersection of red boundary lines in the image) and going as far upstream as is possible on the zoomed image. End by double-clicking.

You needed to zoom in order to avoid conflicts with the auto-snapping feature. If you do click too close to an existing arc you will get a message that the stream is illegal and you will need to try again.

You can end the stream at one location and then continue defining after zooming out by beginning at the point where you left off.

1. Select ***Display*** | ***View*** | ***Previous View***

2. Finish defining each branch. Begin the branch by clicking near the point you left off with and ending by double clicking at the terminal point of the stream.

In order to define separate basins at the junction point you will need to convert the node at the junction to an outlet node.

1. Choose the *Select Feature Point/Node* tool

2. Select the junction point in the center of the watershed corresponding to the intersection of the streams and the sub-basin boundary arcs that you just created.

3. Select ***Feature Objects*** | ***Attributes…***

4. Change the attribute to *Drainage outlet*

5. Select *OK*

6.2.3 Building Polygons

At this point the watershed boundaries are only arcs. In order for them to become polygons you must create the polygon topology.

1. Choose the *Select Polygon* tool

2. Select ***Feature Objects*** | ***Attributes***

3. Change the attribute to *Drainage boundary*. Select *OK*.

4. Choose the *Select Feature Line Branch* tool

5. Select ***Edit*** | ***Select All***

6. While holding down the *Shift* key, select the southern-most stream arc. This un-selects the stream arcs and leaves the remainder selected for use in polygon generation.

7. Select ***Feature Objects*** | ***Build Polygon***

6.2.4 Updating Geometric Parameters

1. Select ***Display*** | ***Display Options…***
2. Turn on the *Color fill polygons* option
3. Select *OK*

In order to transfer the basin area and stream lengths, and to compute them in appropriate units for hydrologic modeling you need to compute the basin data. This will make it possible to use the polygon area in any of the hydrologic modeling interfaces.

1. Select ***Feature Objects*** | ***Compute Basin Data…*** This command computes areas, perimeters, and centroids for each of the sub-basins and assigns these values to the hydrologic modeling tree.
2. In the Units Dialog select the *Current Coordinates…* button
3. Make sure the Horizontal and Vertical units are *Meters* (the base units were UTM meters)
4. Select *OK*
5. Set the *Basin Areas* units to *Square miles*
6. Set the *Distances* units to *Feet*
7. Select *OK* to compute the sub-basin data

6.3 Cleaning

Since you changed some display settings in the last part of the tutorial, you need to completely **exit out of WMS** and **restart the program** before continuing.

1. Switch to the *Map* module
2. Select ***File*** | ***Open…***
3. Open "*streams.img*"

For this file to open properly, *richfield250b.tif* must be located in the same folder as *streams.img*.

1. Choose the *Zoom* tool

2. Using the zoom tool, draw the zoom window shown in Figure 6-2

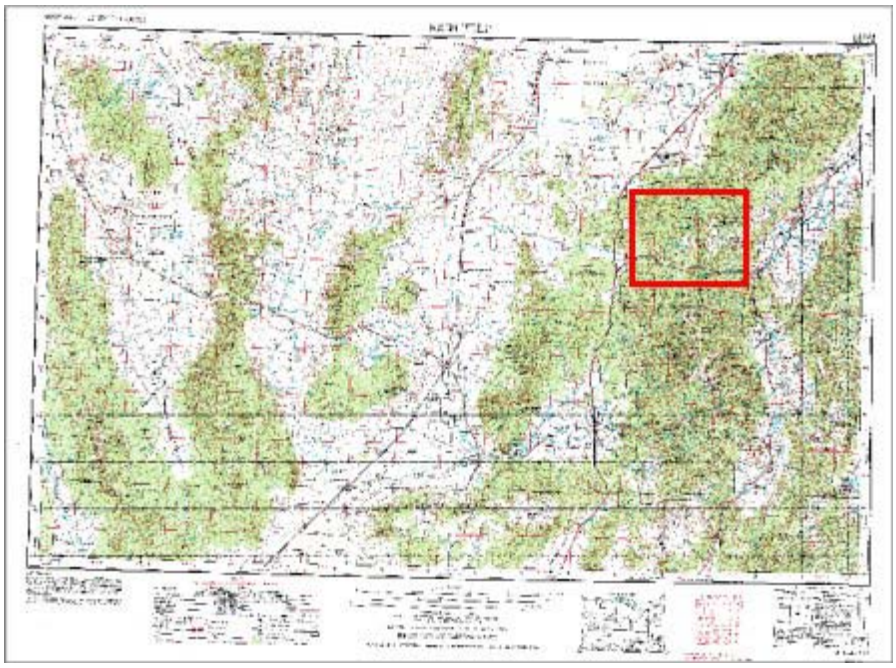


Figure 6-2: Zoom Window

3. As explained in the previous section, digitize the streams and lakes shown in Figure 6-3. Start at the junction the red arrow points to. When you come to a lake, simply trace a stream straight through it.

Figure 6-3: Stream Network and Beginning Junction

6.3.1 Turning Off Image Display

You can now turn off the image by completing the following steps.

1. Expand the *Map Data* folder in the Data Tree if necessary
2. Toggle the visibility check box next to the *"streams.img"* image off

6.3.2 Importing a Shapefile

1. *Select* ***File | Open…***
2. Open "*basins.shp"*
3. Select *OK*

There are some problems with this set of basins and streams. For example, none of the basin junctions meet up with the corresponding stream junctions. Fixing problems like these is called 'cleaning' and is demonstrated in the following steps.

1. Choose the *Zoom* tool
2. Zoom in on the main outlet (at the bottom of the watershed). You will have to get quite close.
3. Choose the *Select Feature Point/Node* tool and select the yellow drainage outlet node .
4. Select ***Feature Objects | Clean…***
5. Make sure the *Snap selected nodes* check box is toggled on
6. Select *OK*

The help window will prompt you to select a snapping point.

1. Click on the bottom vertex of the watershed boundary

Choose the *Frame* macro to view the extents of your model

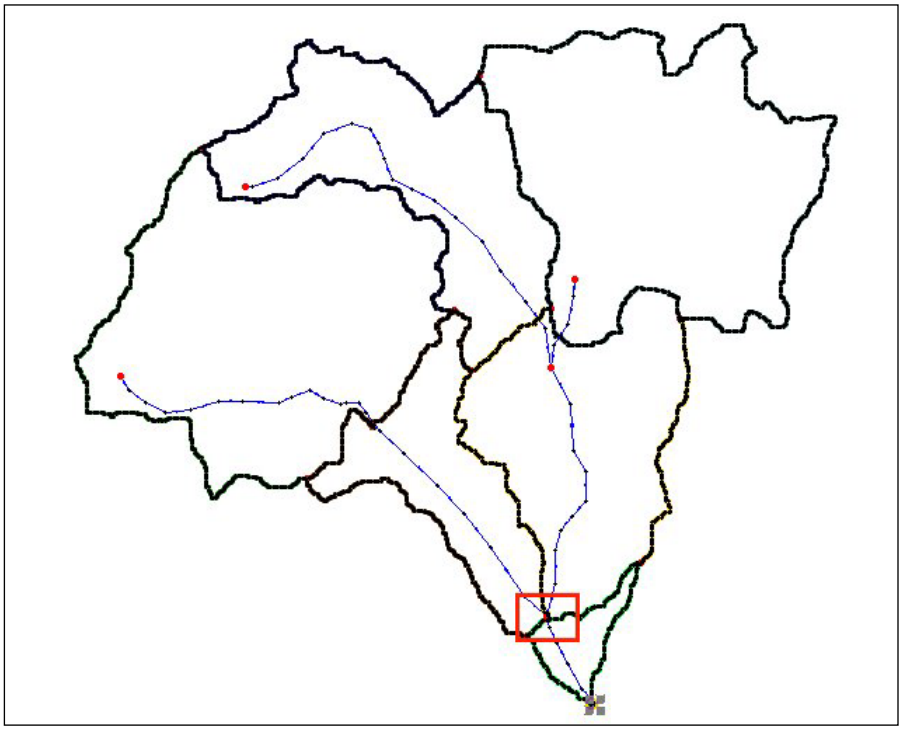


Figure 6-4: Zoom Area for First Upstream Outlet

1. Zoom in around the first upstream interior outlet, as shown in Figure 6-4

2. Choose the *Select Feature Point/Node* tool

3. Select the branching stream node shown in Figure 6-5 Your node may be anywhere, not necessarily above the junction of the colored lines.

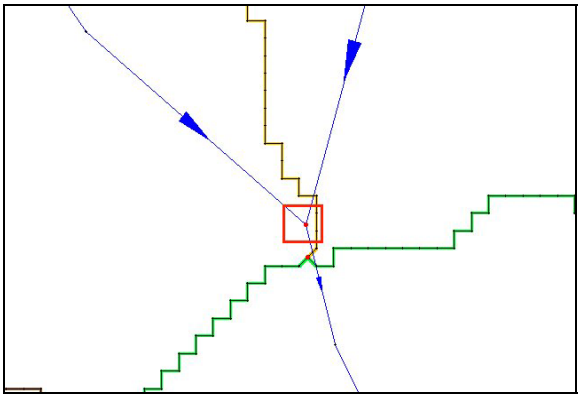


Figure 6-5: Branching Stream Node

1. Select ***Feature Objects | Clean…***
2. Make sure the *Snap selected nodes* checkbox is selected
3. Select *OK*
4. Select the junction of the three basins as the snapping point by clicking on it

This node also needs to be changed to a drainage outlet.

1. Choose the *Select Feature Point/Node* tool
2. Select the node
3. Select ***Feature Objects | Attributes…***
4. Select the *Drainage outlet* radio button
5. Select *OK*
6. Use the *Frame* macro to zoom out
7. Zoom in around the left stream branch and basin junctions, as shown in Figure 6-6

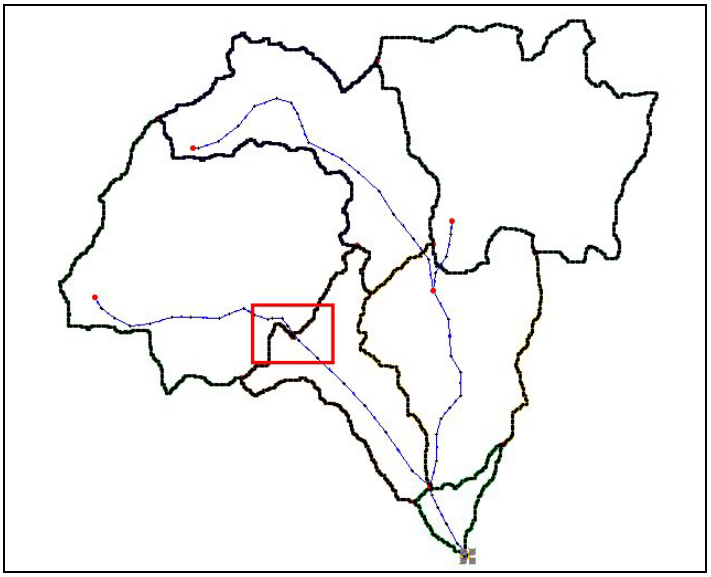


Figure 6-6: Left Stream Branch Zoom Area

1. Choose the *Select Feature Arc* tool

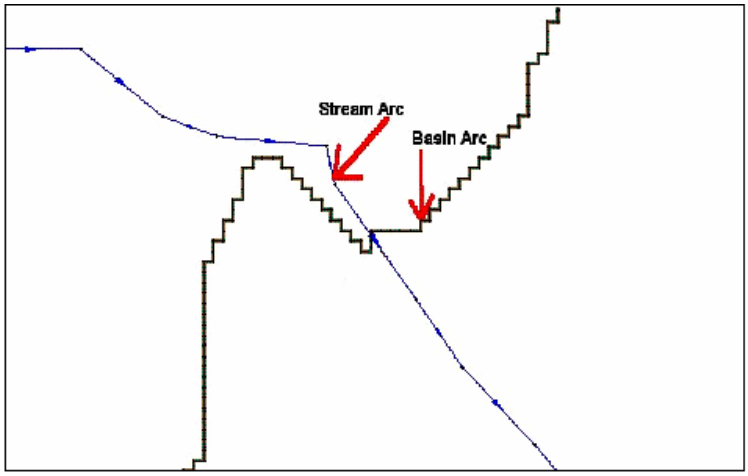


Figure 6-7: Arcs to Clean

1. Select the stream arc and the basin arc it crosses
2. Select ***Feature Objects | Clean…***
3. Make sure the *Intersect selected arcs* check box is selected
4. Select *OK*

The two arcs are intersected and a node is placed at the intersection point. As before, this node needs to be an outlet.

1. Choose the *Select Feature Point/Node* tool
2. Select the node
3. Select ***Feature Objects | Attributes…***
4. Select the *Drainage outlet* radio button
5. Select *OK*
6. Use the *Frame* macro to zoom out
7. Zoom in around the right stream branch and basin junctions as shown in Figure 6-8

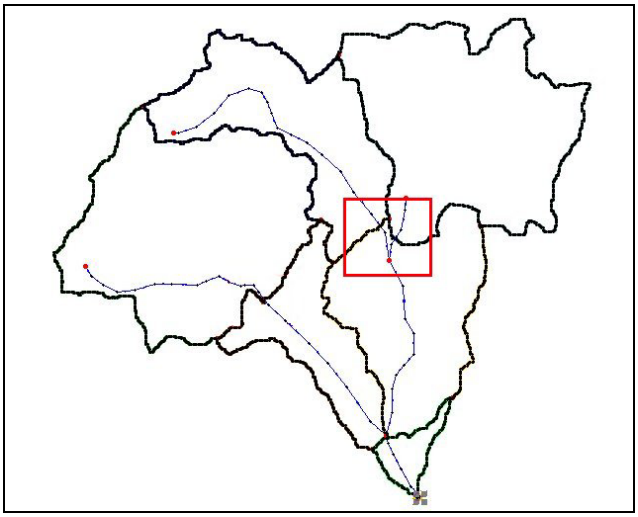


Figure 6-8: Zoom Area for Right Stream Branch

1. Choose the *Select Feature Arc* tool
2. Select the right branch of the stream arc and the basin arc it crosses as shown in Figure 6-9.

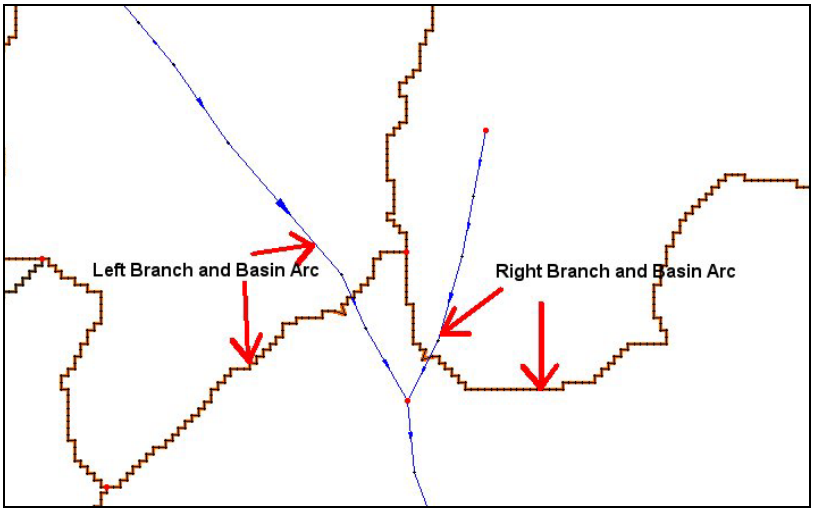


Figure 6-9: Stream Branches and Basin Arcs to Clean

1. Select ***Feature Objects*** | ***Clean…***
2. Make sure the *Intersect selected arcs* check box is selected.
3. Select *OK*.

The two arcs are intersected and a node is placed at the intersection point. As before, this node needs to be an outlet.

1. Choose the *Select Feature Point/Node* tool
2. Select the node
3. Select ***Feature Objects*** | ***Attributes…***
4. Select the *Drainage outlet* radio button
5. Select *OK*
6. Choose the *Select Feature Arc* tool
7. Select the left branch of the stream arc and the basin arc it crosses as shown in Figure 6-9
8. Select ***Feature Objects*** | ***Clean...***
9. Make sure the *Intersect selected arcs* check box is selected
10. Select *OK*

The two arcs are intersected and a node is placed at the intersection point. As before, this node needs to be an outlet.

1. Choose the *Select Feature Point/Node* tool
2. Select the node
3. Select ***Feature Objects | Attributes…***
4. Select the *Drainage outlet* radio button
5. Select *OK*
6. Use the *Frame* macro to zoom out

The set of streams and basins is now cleaned and ready to be used in hydrologic analysis.

6.4 Feature Objects from CAD Data

You may have CAD data available. DWG and DXF data can be automatically converted to feature objects in WMS.

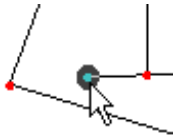
1. Select ***File | New***
2. Select *NO* when asked if you want to save changes
3. Switch to the *Map module*
4. Select ***File | Open…***
5. Open "*af.dwg*"
6. Select ***CAD | CAD -> Feature Objects***

The dialog that opens shows a check mark for each layer that will be converted to feature objects. We will convert all layers and accept the default coverage type (which should be Drainage) and name (which should be CAD layers).

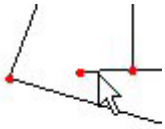
7. Select *OK*
8. Select *OK* to accept the coverage type and name
9. Select ***CAD | Display Options…***
10. Toggle off the check box at the top labeled *Display CAD data*

11. Select *OK*

Because these lines were created in AutoCAD we can't be sure (and in most cases they won't) that the streams are created using the WMS conventions for direction. In order to fix any such problems you can use the Reorder Streams command. By selecting the most downstream node in a stream network and invoking the Reorder Streams command, you tell WMS to ensure that all arcs are ordered downstream to upstream from the selected point.



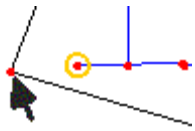
1. Choose the *Select Feature Point/Node* tool
2. Select the left-most node in the interior of the basin (the left-most node on the portion that forms a network inside of the arcs forming a boundary).
3. Select ***Feature Objects*** | ***Reorder Streams***



4. Choose the *Select Feature Line Branch* tool
5. Select the arc attached to the left-most node (the node that was just used to reorder streams)
6. Select ***Feature Objects*** | ***Attributes…***
7. Select the *Stream* type and Select *OK*

Each stream now flows the proper direction, toward the one drainage outlet at the left of the stream network. This outlet needs to be snapped to the basin boundary.

1. Choose the *Select Feature Point/Node* tool
2. Select the drainage outlet
3. Select ***Feature Objects*** | ***Clean…***
4. Make sure *Snap selected nodes* is checked.
5. Select *OK*



6. Choose the node on the basin boundary closest to the drainage outlet.
7. Select ***Feature Objects*** | ***Build Polygon***
8. Select *OK*

This set of streams and basins is now properly ordered and connected and is ready to be used for hydrologic analysis.

6.5 Conclusions

In this workshop you should have learned how to do the following:

8. Use feature object drainage coverages for watershed delineation
9. Advanced feature object editing
10. Assign appropriate feature object attributes

CHAPTER 7

DEM Delineation

Watershed delineation from DEMs is straightforward and relatively simple, provided the project area is not entirely flat or completely dominated by man-made structures (you can't expect the DEM method to work if there is no relief in the DEM elevations themselves).

Because some DEMs do not contain enough resolution to accurately incorporate roadways, canals, and other man-made structures in the delineation, you may need to alter the delineation either by using additional stream arcs, or by editing the elevations. This process, including how to alter delineations when necessary is the subject of this workshop.

7.1 Objectives

In this workshop you will learn the basics of importing, viewing, and preparing DEMs for automated watershed delineation. This includes the following:

1. Computing flow paths and flow accumulations
2. Delineating watersheds from DEMs
3. Delineating sub-basins within a watershed
4. Using feature line streams to modify delineations

7.2 Importing DEM Data

The first step in delineating a watershed is to import one or more DEMs.

To read in a set of eight 30-meter DEMs from the 1:24000 series, complete the following steps:

1. Select ***File*** | ***Open…***
2. Open "*covefort.dem*"
3. In the *Importing USGS DEMs* dialog, click the *Add* button
4. Add the following seven DEMs just as you did in the previous tutorial:
   - josephpeak.dem
   - marysvalecanyon.dem
   - mountbelknap.dep
   - mountbrigham.dem
   - polemountain.dem
   - redridge.dem
   - trailmountain.dem
5. Enter a thinning factor of **3** in the *Thinning factor* edit field

Thinning the resolution of the DEMs will reduce the density of elevation points so that your computer will process the DEM data faster. The resolution of points in the 30-meter DEMs is too dense for the purposes of this tutorial, so you will not lose any accuracy by thinning.

1. Select *OK*

7.2.1 Trimming the DEM

1. Select ***DEM*** | ***Trim*** | ***Polygon…***
2. Select the *Enter Polygon Interactively* option
3. Select *OK*
4. Trace around the area formed by the polygon shown in Figure 7-1

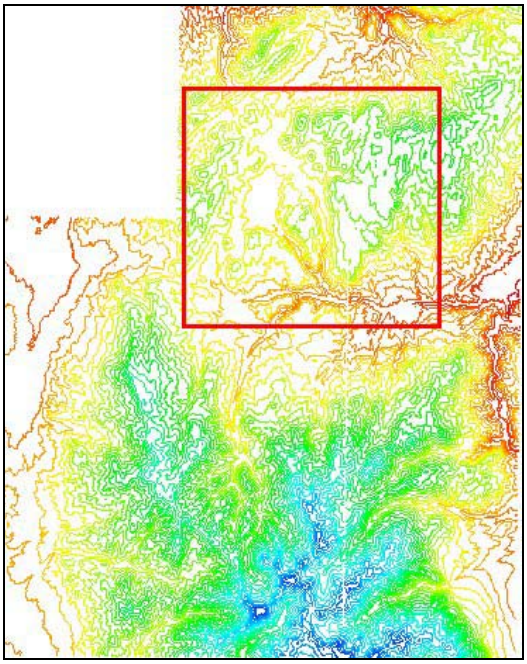


Figure 7-1: DEM Trim Area

5. Select ***DEM*** | ***Fill*** to linearly interpolate and fill gaps that were created by thinning the DEM data

6. Select *OK*

7.3 Computing Flow Data and Accumulations

1. Switch to the *Drainage* module

2. Select ***DEM*** | ***Compute TOPAZ Flow Data***

3. Select *OK*

4. Select *OK*

5. Once TOPAZ finishes running (you may have to wait a few seconds to a minute or so), choose *Close* to close the TOPAZ window

You should now see a network of streams on top of your DEM. TOPAZ computes flow directions for individual DEM cells and creates streams based on these directions. To learn more about this process, consult the WMS Help File. You can change the flow accumulation threshold so that smaller or larger streams show up.

1. Choose the *Frame* macro

2. Select ***Display*** | ***Display Options…***
3. Select the *DEM* tab
4. Change the *Min Accumulation For Display* to **0.2** and select *OK*

You should now see a more detailed network of streams.

1. Select ***Display*** | ***Display Options…***
2. Select the *DEM* tab
3. Change the *Min Accumulation For Display* to **1.5** and select *OK*

You should now see only a few large stream branches.

7.3.1 Drawing Flow Paths

WMS can draw a flow path from any point on a watershed to the eventual outlet.

1. Choose the *Flow Path* tool
2. Click on any portion of the DEM and see the resulting flow path
3. After you have drawn a few flow paths, choose the *Refresh* tool to clean up the image.

7.4 Delineating Watersheds from DEMs

WMS uses the data computed by TOPAZ to delineate watershed basins.

7.4.1 Creating the Basin Outlet

1. Zoom in around the area shown in Figure 7-2.

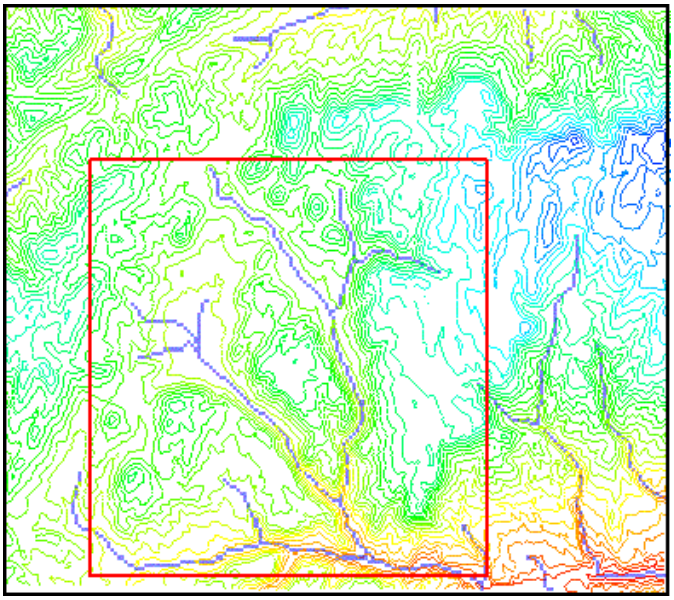


Figure 7-2: DEM Zoom Area

1. Choose the *Create Outlet Point* tool

2. Place the node just upstream of the branch junction shown in Figure 7-3

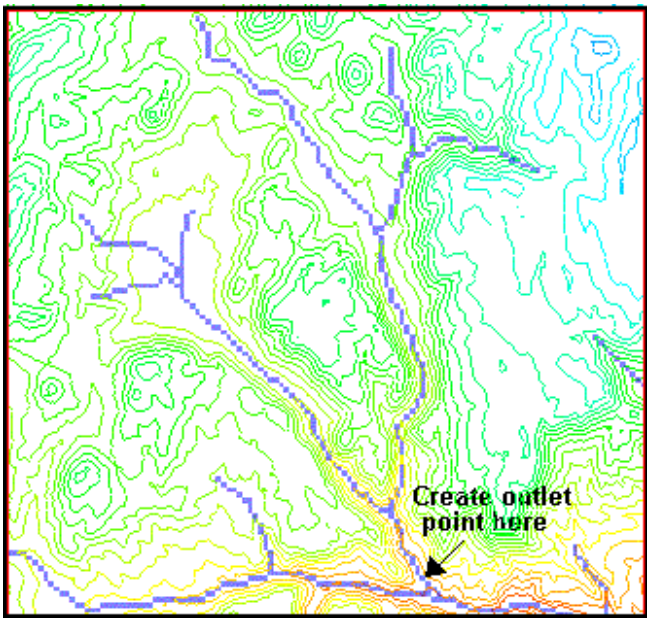


Figure 7-3: Drainage Outlet Location

1. Choose the *Frame* macro

2. Select ***DEM | DEM -> Stream Arcs***

3. Turn on both the *Display stream feature arc creation* option and the *Use feature points to create streams* options
4. Set the *Threshold value* to **0.3**
5. Select *OK*.

By creating a point on a lower stream branch, you defined a watershed outlet. WMS then used the flow direction and accumulation data from TOPAZ to convert the TOPAZ streams into stream feature arcs.

7.4.2 Defining a Basin

These stream feature arcs computed by TOPAZ can now be used to define basins.

1. Select ***DEM*** | ***Define Basins***

Now that the basins have been delineated, you can convert them to feature polygons and compute the basin data.

1. Select ***DEM*** | ***Basins->Polygons***
2. Select ***DEM*** | ***Compute Basin Data…***
3. Select *OK* on the *Units* dialog
4. Select ***Display*** | ***Display Options…***
5. Select the *Drainage Data* tab
6. Check the *Show Units*, *Basin Areas*, and *Basin Slopes* options
7. Select *OK*.

Using DEM data, you have delineated a watershed and stream network. Because the DEM contains elevation data, your model can be used to compute data such as slopes, stream lengths, areas, etc. It is not just a schematic of the watershed.

7.5 Creating Sub-Basins

In previous watershed models you have looked at, each watershed was divided into several smaller sub-basins. You will learn how to create these interior sub-basins in this part of the tutorial. You should always finish delineating the major watershed as shown in parts 7.3 and 7.4 of this tutorial before attempting to create sub-basins.

7.5.1 Adding Drainage Outlets

In order to create sub-basins, you can convert existing nodes and vertices along the stream arcs into drainage outlets.

1. Zoom in around the area shown in Figure 7-4

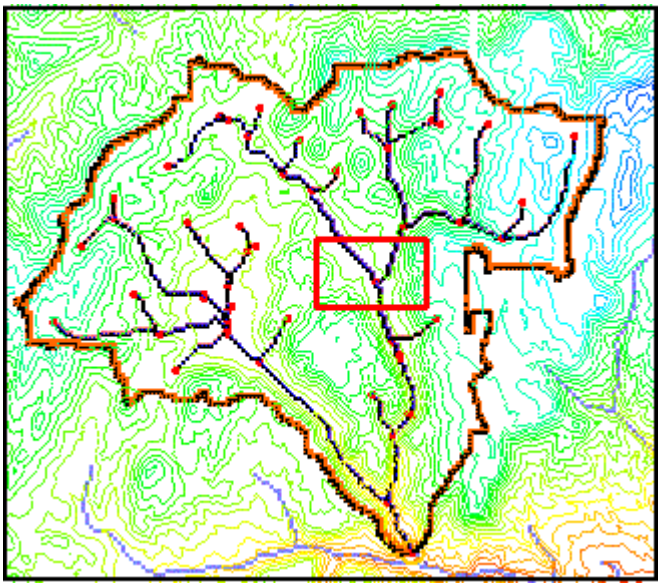


Figure 7-4: Zoom Area

1. Choose the *Select Feature Vertex* tool

2. Select the vertex at the location shown in Figure 7-5

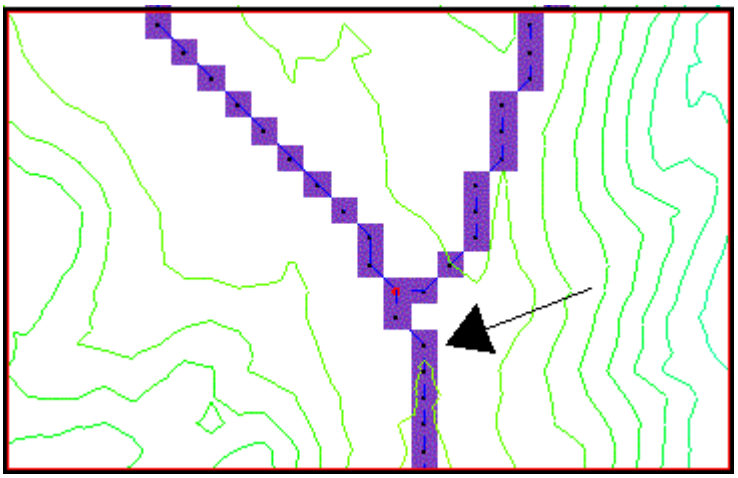


Figure 7-5: Node Location

1. Select ***DEM | Node<->Outlet***

2. Choose the *Frame Image* macro
3. Zoom in around the branch shown in Figure 7-6

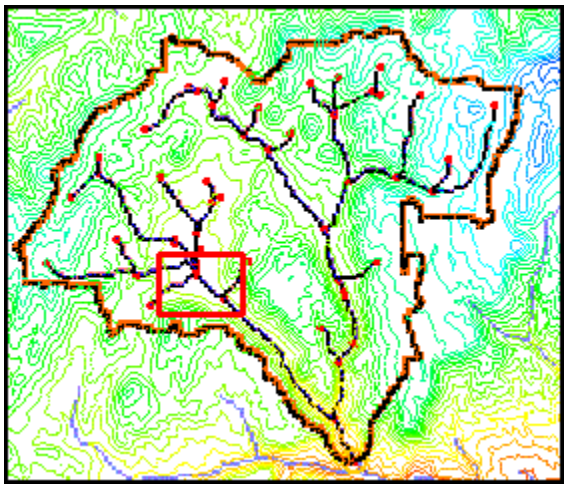


Figure 7-6: Zoom Area

1. Choose the *Select Feature Point/Node* tool
2. Double-click on the most downstream red node you can see, as shown in Figure 7-7

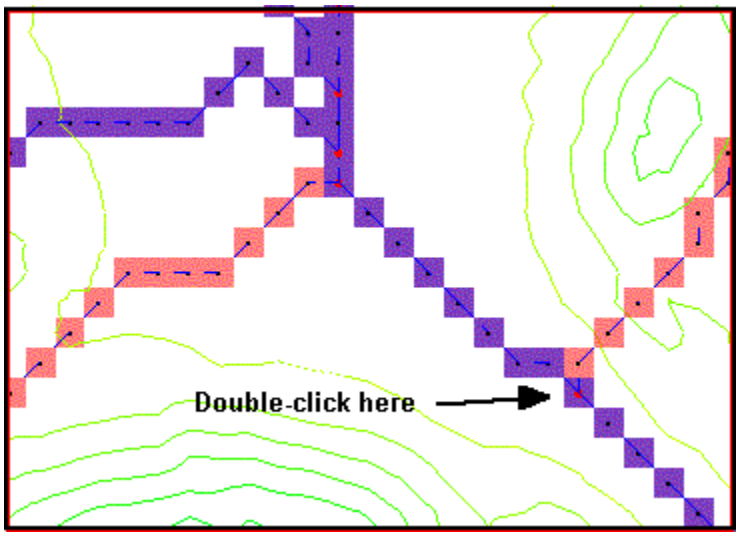


Figure 7-7: Change the indicated node to an outlet.

3. Select the *Drainage outlet* option
4. Select *OK*

5. Select the *Frame Image* macro

7.5.2 Defining Sub-Basins

1. Select ***DEM | Define Basins***
2. Select ***DEM | Basins->Polygons***
3. Select ***DEM | Compute Basin Data…***
4. Select *OK*

Notice that on the left basin, where you changed the existing node to an outlet, the basin was divided into two smaller sub-basins automatically. On the right basin, where you added an outlet below the stream junction, only one basin was created.

7.6 Adding a Stream Arc and Redefining Basins

Sometimes you will need to add stream arcs to your basin to represent man-made objects such as roads. Roads often disrupt the natural flow of watersheds and water collects along roads just as it collects in a stream. This collected water needs to be "added" into your watershed in order to properly model the real-life situation.

1. Select ***File | Open…***
2. Open "*richfield250a.tif*"
3. Switch to the *Map* module
4. Choose the *Zoom* tool
5. Zoom in along the road as shown in Figure 7-8.

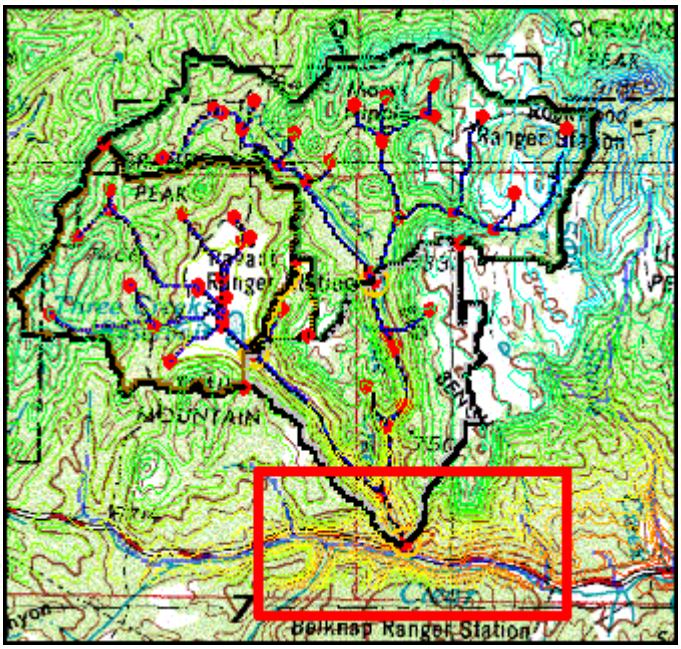


Figure 7-8: Road Zoom Area

1. Choose the *Create Feature Arc* tool
2. Select ***Feature Objects*** | ***Attributes…***
3. Select the *Streams* option.
4. Select *OK*
5. Create the arcs as shown in Figure 7-9. Begin the first arc by clicking somewhere on the existing stream arc. Then continue the arc along the roadway to the left. Begin the second arc at the same beginning point of the first arc and continue along the road to the right.

Make sure both arcs are created so that the existing drainage outlet is downstream. Both stream arcs should empty into the stream above the existing outlet. Also, to avoid creating basins on the south side of the highway, draw each arc on the north side of the flow accumulation lines. You are creating the stream arcs along the road to show that the water collected by the road drains into the same area the main watershed drains into.

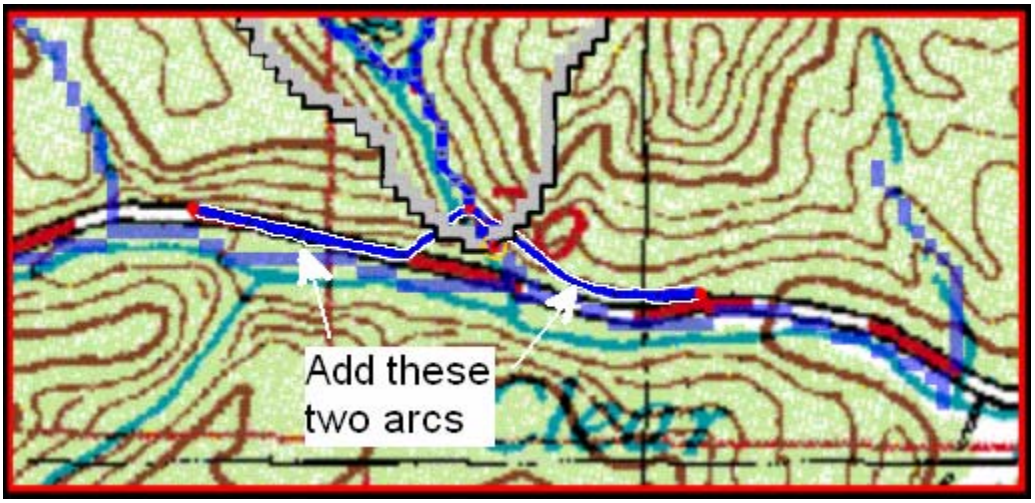


Figure 7-9: Roadway Collection Stream Arcs

You may want to analyze the flow along the road as part of the flow in the original basin.

1. Switch to the *Drainage* module
2. Select ***DEM | Define Basins***
3. Select ***DEM | Basins->Polygons***
4. Select ***DEM | Compute Basin Data…***
5. Select *OK*

You can see that the drainage along the road has now been diverted into the original basin. Your basins should look something like Figure 7-10.

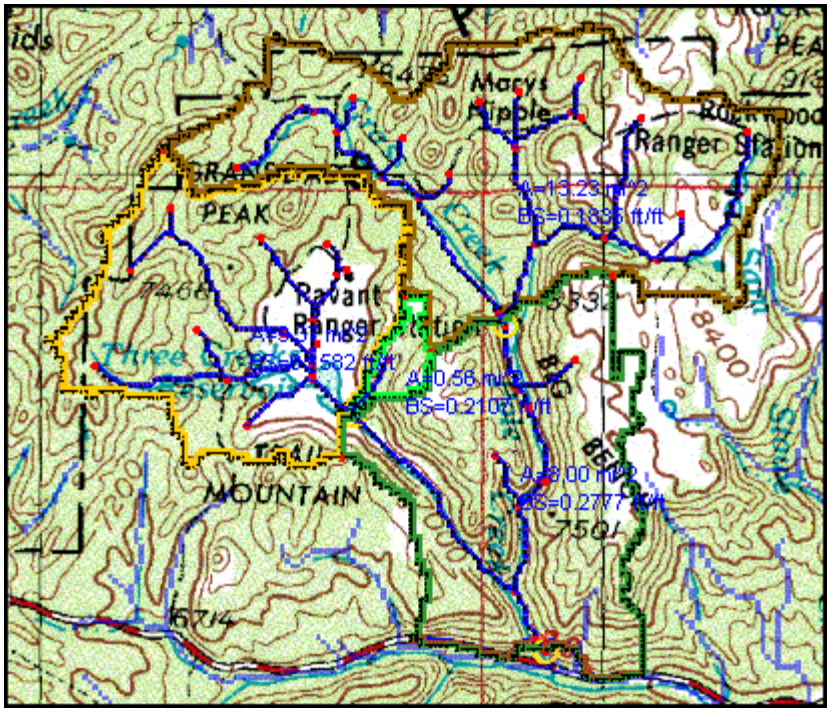


Figure 7-10: New Basins from Road Stream Arcs

Now you will change the node where the new streams enter the original stream to be an outlet. Basins will be created for the new stream arcs and the original stream.

1. Choose the *Select Nodes* tool
2. Select the node on the original stream that was created when you made the road stream arcs
3. Select ***DEM | Node<->Outlet***
4. Select ***DEM | Define Basins***
5. Select ***DEM | Basins->Polygons***
6. Select ***DEM | Compute Basin Data…***
7. Select *OK*

If you want to combine everything into one basin again, you can change the outlet back to a node and redefine basins.

7.7 Displaying DEMs

WMS has several options for displaying DEMs.

7.7.1 Move Basin Labels

You may want to move the basin labels displaying the area, slope, etc. to more convenient locations on the image.

1. Choose the *Move basin label* tool
2. Click on any area in the basin you just selected and drag the mouse to a location outside the basin before letting go of the mouse button

An arrow will be drawn from where you let go of the mouse to where you started it.

7.7.2 Changing Text Attributes

1. Select ***Display*** | ***Display Options…***
2. Choose the *Drainage Data* tab
3. Select the color box next to *Data text color*

You can change the text attributes, such as color, font, and fill behind color in this dialog box.

1. Once you are done, Select *OK* to both dialogs

7.7.3 Color Filling Basins

WMS also allows you to fill in each basin with a different color. This is useful when you do not need the background image to show up.

1. Select the *Frame Image* macro
2. Select ***Display*** | ***Display Options…***
3. Choose the *Map* tab
4. Check the *Color fill polygons* option
5. Select *OK*

7.7.4 Turning Displays Off

When you are finished using the elevation data, you may want to turn the DEM contours and extra streams displays off.

1. Select ***Display*** | ***Display Options…***
2. Choose the *DEM* tab
3. Uncheck the *Flow accumulation* and *DEM Contours* options
4. Select *OK*

7.7.5 Smoothing Boundaries

If you zoom in near the boundary of one of your basins, you will see that the boundary lines are not smooth. WMS allows you to redistribute vertices to smooth these boundaries.

1. Switch to the *Map* module
2. Choose the *Select Feature Arcs* tool
3. Select ***Edit*** | ***Select All***
4. Select ***Feature Objects*** | ***Redistribute…***
5. Select the *Use Cubic Spline* option
6. Enter **100** in the *Spacing* edit box
7. Select *OK*

When you zoom in on the basin boundaries now, they should be much smoother.

7.7.6 Shading

As you learned in the last tutorial, WMS has several options for shading DEMs.

1. Select ***Display*** | ***Shading Options***

This time the basin boundaries should show up in color on the shaded image. Remember that you can change the position of the light source and other shading options to achieve different effects.

7.8 Conclusions

In this workshop you should have learned how to use DEMs data in WMS. This includes the following:

1. Computing flow paths and flow accumulations
2. Delineating watersheds from DEMs
3. Delineating sub-basins within a watershed
4. Using feature line streams to modify delineations

CHAPTER 8

Editing DEMs

Some terrain features, including man-made features such as roads, canals, reservoirs, dams, or dikes and levees, may not be well represented if the resolution of the DEM is coarse. Evaluating future changes in terrain may also be desirable. It is possible to edit DEM elevations and flow directions in order to include these features in the surface representation.

Storage capacity curves can be computed based on a DEM and then used for reservoir routing in the Detention Basin calculator.

8.1 Objectives

In this tutorial you will learn how to edit DEMs for more accurate surface representation and drainage analysis by completing the following steps:

1. Delineating a basin using TOPAZ flow accumulations and directions
2. Filling gaps of data
3. Editing flow directions
4. Editing elevations to create streams
5. Editing elevations using feature arcs
6. Computing a storage capacity curve for a reservoir/basin

7. Routing an input hydrograph through a reservoir/basin

8.2 Running TOPAZ and Basin Delineation

Using the flow accumulations and flow directions that TOPAZ outputs for each DEM cell makes basin delineation very simple.

8.2.1 Open DEMs

1. Select ***File*** | ***Open…***
2. Open "*mvcanyon.dem"*and *"trailmount.dem"*
3. Select *Open*
4. Select *OK*

8.2.2 Run TOPAZ

1. Switch to the Drainage module
2. Select ***DEM*** | ***Compute TOPAZ Flow Data…***
3. Select *OK*
4. Select *OK*
5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)
6. Select ***Display*** | ***Display Options…***
7. Change the *Minimum Accumulation For Display* to **0.06** mi^2
8. Select *OK*
9. Zoom in to the top middle section of the DEM as shown in Figure 8-1

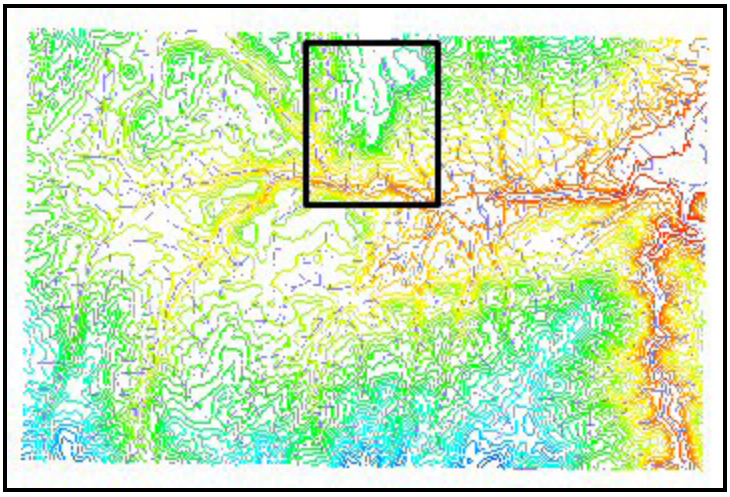


Figure 8-1: Zoom in to the rectangle on the DEM

8.2.3 Basin Delineation

1. Select the *Create Outlet Point* tool
2. Click anywhere on the DEM to create an outlet
3. Select *OK* if you get a message telling you that the outlet is not located in a flow accumulation cell
4. Enter an X-value of **379589.5** and a Y-value of **4271008.5** in the Edit Window at the top of the screen to edit the outlet location
5. Select ***DEM | Delineate Basins Wizard***
6. Select *OK*
7. Select *OK*

8.3 DEM Fill Command

The delineation of this basin, shown in Figure 8-2 looks a little bit suspicious. First of all, the upper-right edge of the basin boundary is linear and flat. This usually indicates that the basin actually continues further, but was not properly delineated for some reason. The other noticeable item is the gap in the flow accumulation cells. It looks like the stream does not totally connect. This is indicative of gaps in our data. The DEM Fill command is useful for filling gaps in DEM data. We will identify the cause of the delineation error, use the DEM Fill command, recompute flow accumulations and directions, and delineate the correct basin.

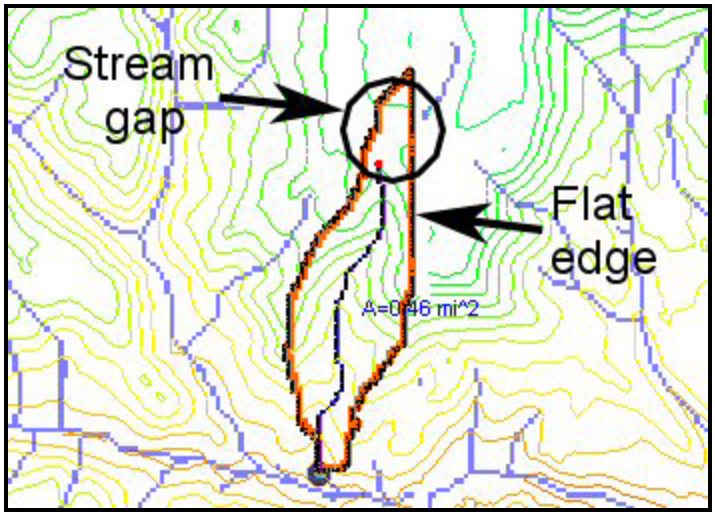


Figure 8-2: Delineation errors

8.3.1 Basin Delineation Errors

1. Select ***Display*** | ***Display Options…***
2. Toggle *No Data Cells* on
3. Select *OK*
4. There are some No Data cells that interfere with the basin delineation as shown in Figure 8-3

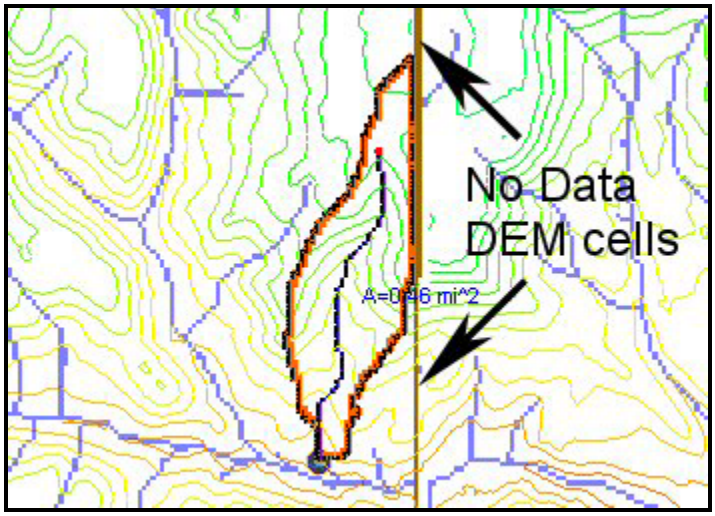


Figure 8-3: No Data cells cause incorrect basin delineation

5. Select ***Display*** | ***Frame Image***

6. Switch to the Terrain Data module

7. Select ***DEM | Fill***

8. Select *OK*

Notice that values were interpolated for the No Data cells lying in the interior part of the DEM.

8.3.2 Run TOPAZ

1. Switch to the Drainage module

2. Select ***DEM | Compute TOPAZ Flow Data…***

3. Select *OK*

4. Select *OK*

5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

6. The flow accumulation cells now connect all the way through as shown in Figure 8-4

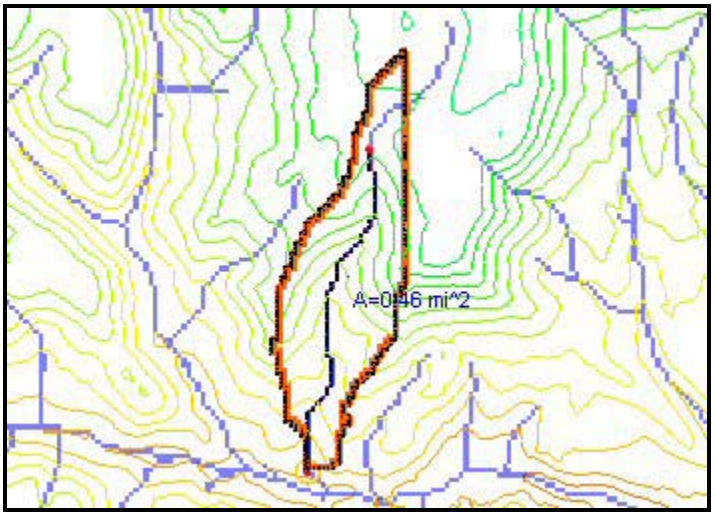


Figure 8-4: New TOPAZ results after using the DEM | Fill command

8.3.3 Basin Delineation

1. Select ***DEM*** | ***Delineate Basins Wizard***
2. If prompted, select *OK* to delete all existing feature data and recreate it using the new basin delineation
3. Select *OK*
4. Select *OK*

Figure 8-5 shows the corrected basin delineation.

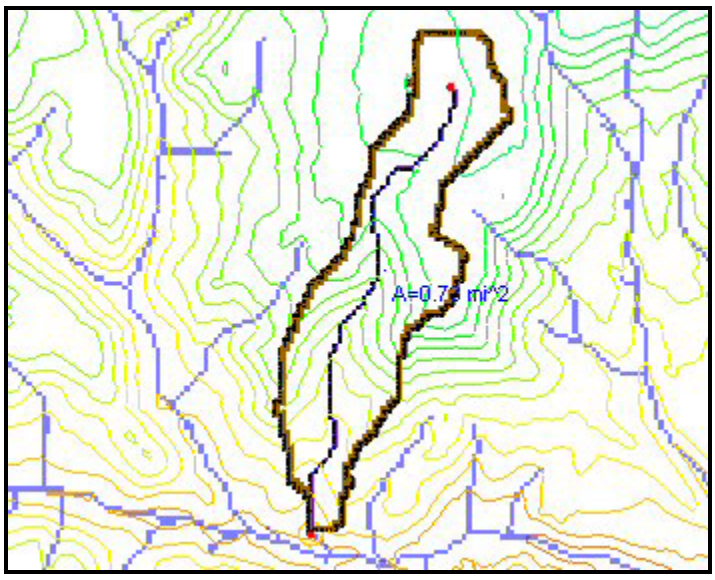


Figure 8-5: Final basin delineation

8.4 Editing Flow Directions

Flow directions can be inaccurate due to imprecision in the DEM. The flow direction in each DEM cell can be manually edited in order to improve accuracy in basin delineation.

8.4.1 Open DEM

1. Select ***File*** | ***New***
2. Select *NO* when asked if you want to save your changes
3. Select ***File*** | ***Open…***

4. Open *"trailmount.dem"*
5. Select *OK*

8.4.2 Open Image

1. Select ***File*** | ***Open…***
2. Open *"trailmountain.TIF"*
3. Zoom in to the area in the rectangle of the DEM as shown in Figure 8-6

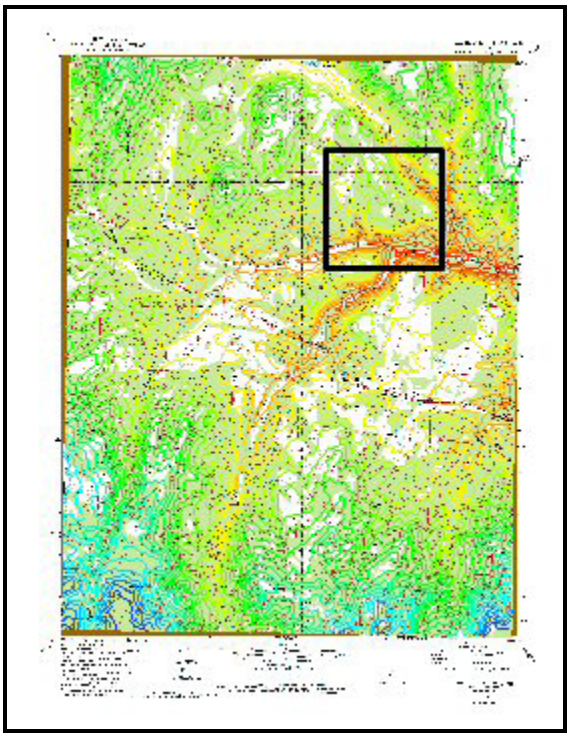


Figure 8-6: Zoom in to rectangle

8.4.3 Run TOPAZ

1. Switch to the Drainage module
2. Select ***DEM*** | ***Compute TOPAZ Flow Data…***
3. Select *OK*
4. Select *OK*

5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

Notice that the flow accumulations show a stream that differs from the stream shown in the background image in Figure 8-7.

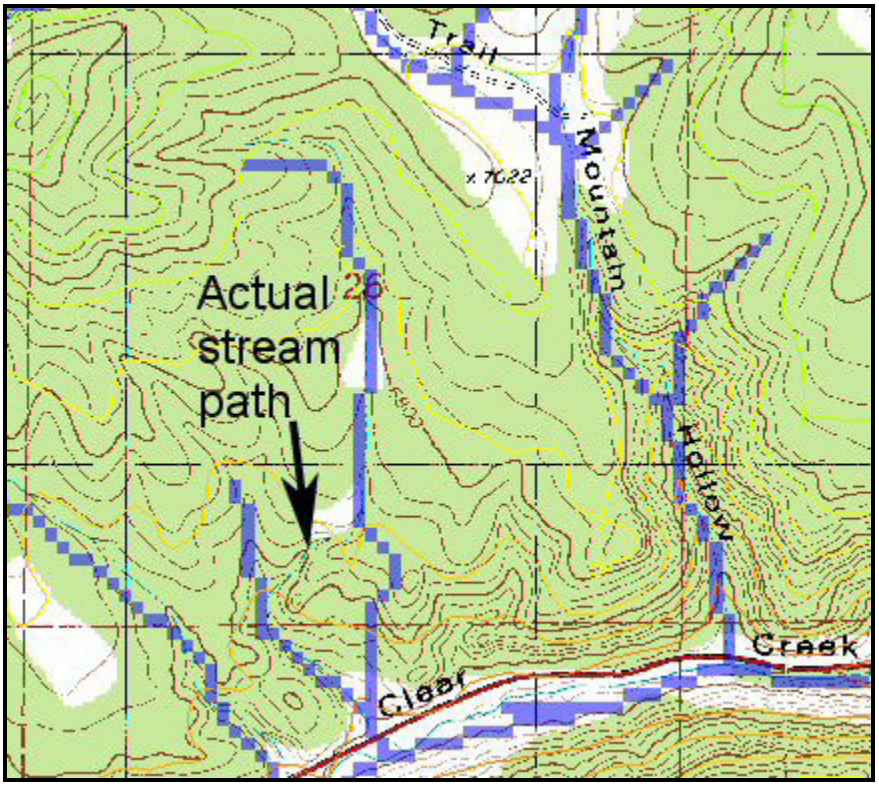


Figure 8-7: Stream path differences

8.4.4 Edit Flow Directions

Correct the flow directions for each of the numbered DEM cells in Figure 8-8.

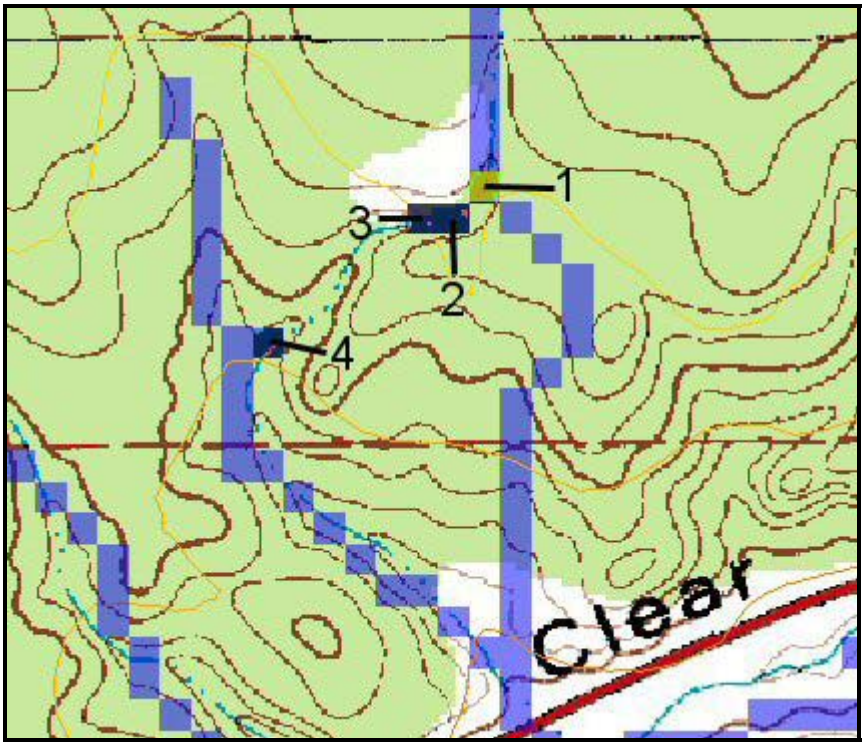


Figure 8-8: Edit flow directions for the numbered DEM cells

1. Use the *Select DEM points* tool and double-click on a numbered cell

This will bring up the DEM Point Attributes dialog shown in Figure 8-9.

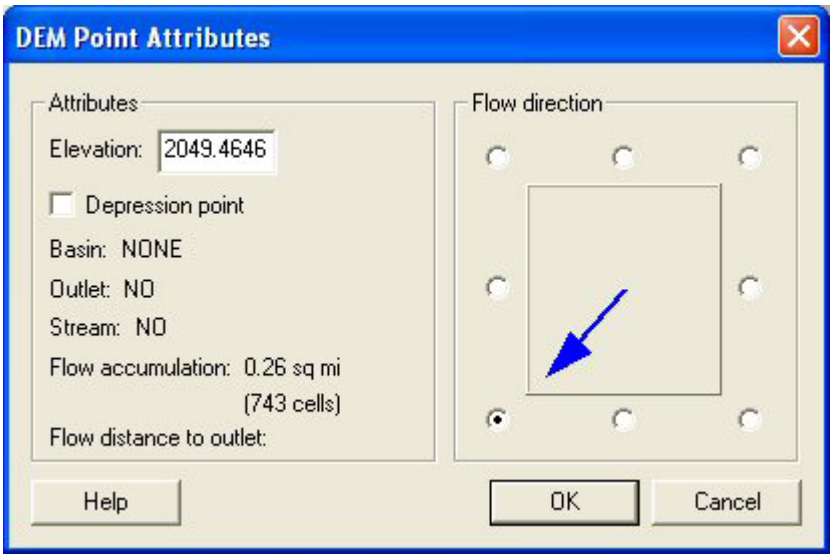


Figure 8-9: DEM Point Attributes dialog

2. Change the flow direction according to Table 8-1

Table 8-1: New flow directions

DEM Cell	New Flow Direction
1	↙
2	←
3	←
4	↙

3. Select *OK*

4. Toggle the *Compute flow accumulations* on ONLY after the last flow direction has been edited

5. Select *OK*

6. Repeat Step 1-5 for all remaining numbered DEM cells in Figure 1-8

It is possible to edit the flow direction for multiple DEM cells at once using the ***DEM | Point Attributes*** command in the Map module

8.4.5 Delineate Basin

1. Use the *Create outlet point* tool to create an outlet point as shown in Figure 8-10

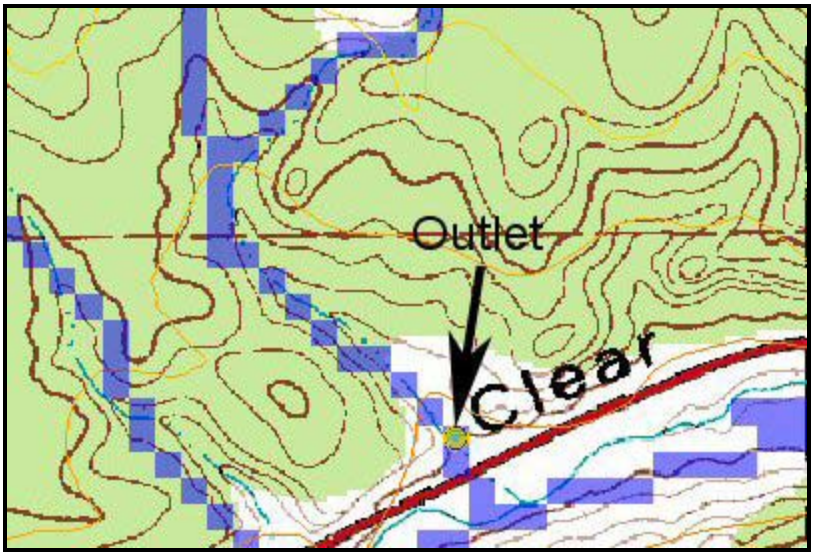


Figure 8-10: Outlet location

2. Select ***DEM | Delineate Basins Wizard***
3. Select *OK*
4. Select *OK*

8.5 Editing Elevations to Create Streams

Stream arcs on a DEM are generally created based on flow directions and flow accumulations. DEM elevations do not always represent the channel elevation of a stream, but might actually be the bank elevation. This can cause streams that have unnatural profiles with drastically varying slopes. We will smooth the stream cell profiles to be more representative of a natural sloping channel.

8.5.1 Editing Elevations Using Stream Arcs

1. Switch to the Terrain Data module
2. Select ***Display | Display Options…***
3. Toggle *Stream*, *Flow Accumulation, Color Fill Drainage Basins,* and *Fill Basin Boundary Only* off
4. On the Map tab change the *Points/Nodes* and *Vertices* Radius to **2** by clicking on their corresponding buttons on the left
5. Select *OK*

6. Select *OK*

7. Use the *Select Feature Arc* tool to select the arc shown in Figure 8-11

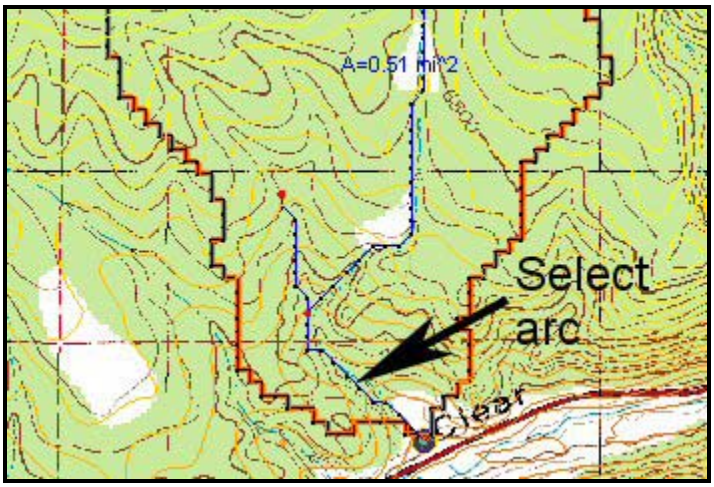


Figure 8-11: Select stream arc

8. Select ***DEM | Edit Elevations***

You should see the profile of the DEM elevations underlying the stream arc as shown in Figure 8-12. The elevations are displayed downstream to upstream as you move left to right. There is an unnatural depression in the stream that is obvious when looking at the profile of the elevations. We could use the Interpolate button to smooth the entire stream to a constant slope but doing this is too much of a generalization. It is better to break the arc into multiple arcs, each arc being characterized by a similar slope, for interpolation.

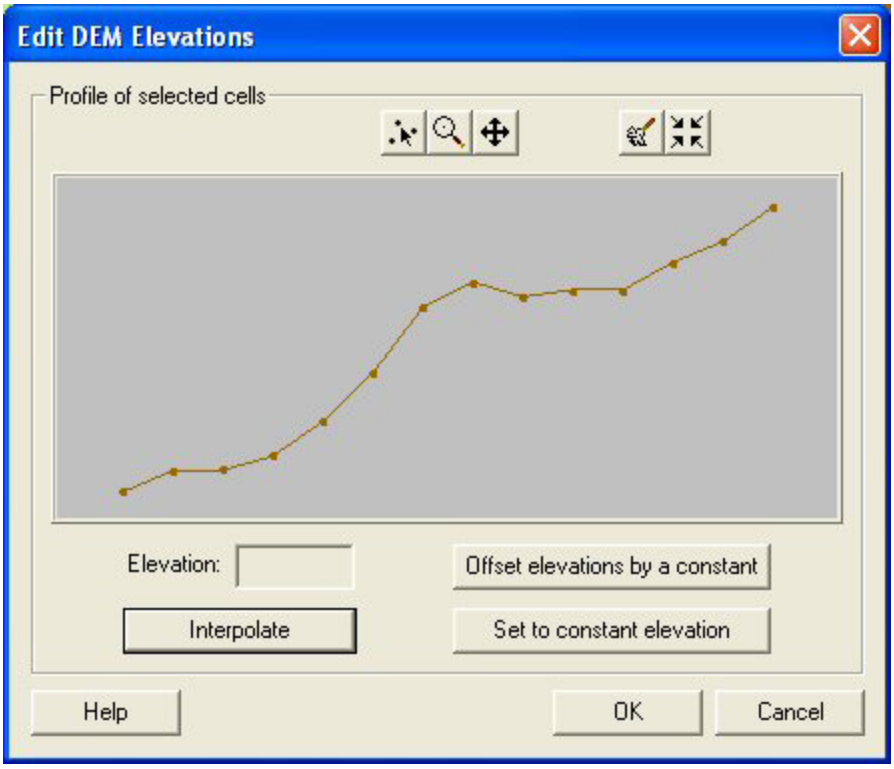


Figure 8-12: Stream arc profile

9. Select *Cancel*

8.5.2 Convert Arc Vertices to Nodes

Converting vertices to nodes will break the original arc into multiple arcs.

1. Switch to the Map module
2. Use the *Select Feature Vertex* tool to select the vertices shown in Figure 8-13

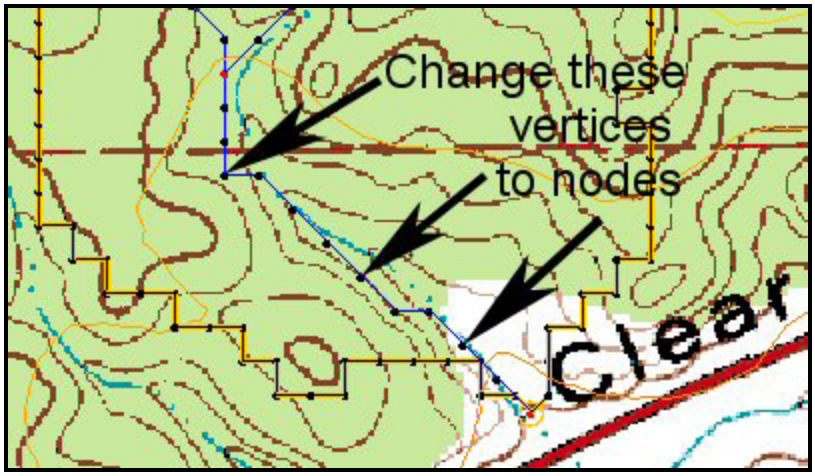


Figure 8-13: Changing vertices to nodes

3. Select ***Feature Objects | Vertex <-> Node***

8.5.3 Interpolate Elevations

1. Switch to the Terrain Data module
2. Use the *Select Feature Arc* tool to select the first numbered arc in Figure 8-14

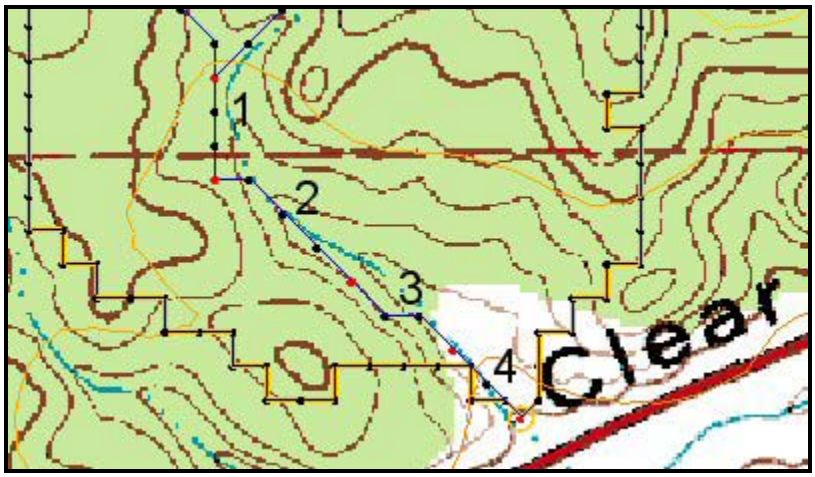


Figure 8-14: Interpolation arcs

3. Select ***DEM | Edit Elevations***
4. Select the *Interpolate* button
5. Select *OK*

6. Repeat the same process for each of the numbered arcs

8.5.4 Convert Arc Nodes to Vertices

Convert the nodes that were created in Section 8.5.2 back to vertices in order to offset the entire arc at once.

1. Switch to the Map module
2. Use the *Select Feature Point/Node* tool to select the nodes that were converted from vertices using Figure 8-13
3. Select ***Feature Objects | Vertex <-> Node***

8.5.5 Offset Elevations

Offsetting the elevations of smoothed cells along the stream arc will make a definite stream in the DEM.

1. Switch to the Terrain Data module
2. Use the *Select Feature Arc* tool to select the original arc shown in Figure 8-11
3. Select ***DEM | Edit Elevations***
4. Select the *Offset elevations by a constant* button
5. Enter **-2**
6. Select *OK*
7. Select *OK*

8.5.6 Run TOPAZ

1. Switch to the Drainage module
2. Select ***DEM | Compute TOPAZ Flow Data…***
3. Select *OK*
4. Select *OK*

5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

8.5.7 Edit Flow Directions

1. Select ***Display*** | ***Display Options…***
2. Toggle *Flow Accumulation* on
3. Select *OK*
4. Repeat steps in Section 8.4.4 to edit flow directions

8.5.8 Delineate Basin

1. Select ***DEM*** | ***Delineate Basins Wizard***
2. If prompted, select *OK* to delete all existing feature data and recreate it using the new basin delineation
3. Select *OK*
4. Select *OK*

8.6 Editing Elevations Using Feature Arcs

DEM cell elevations can be edited to more accurately model the terrain or to reflect man made structures such as roads, dikes or levees, dams, reservoirs, or detention basins. This makes creating a terrain model very versatile and allows you to evaluate a number of different scenarios.

8.6.1 Prepare to Edit Elevations

1. Zoom in around the outlet
2. Hide *"trailmountain.tif"* by toggling its visibility checkbox off
3. Select ***Display*** | ***Contour Options…***
4. For the *Contour Interval* change the *Number of contours* to **100**
5. Select *OK*
6. Switch to the Terrain Data module

8.6.2 Edit Elevations Using an Generic Arc

1. Use the *Create Feature Arc* tool to strike an arc as shown in Figure 8-15

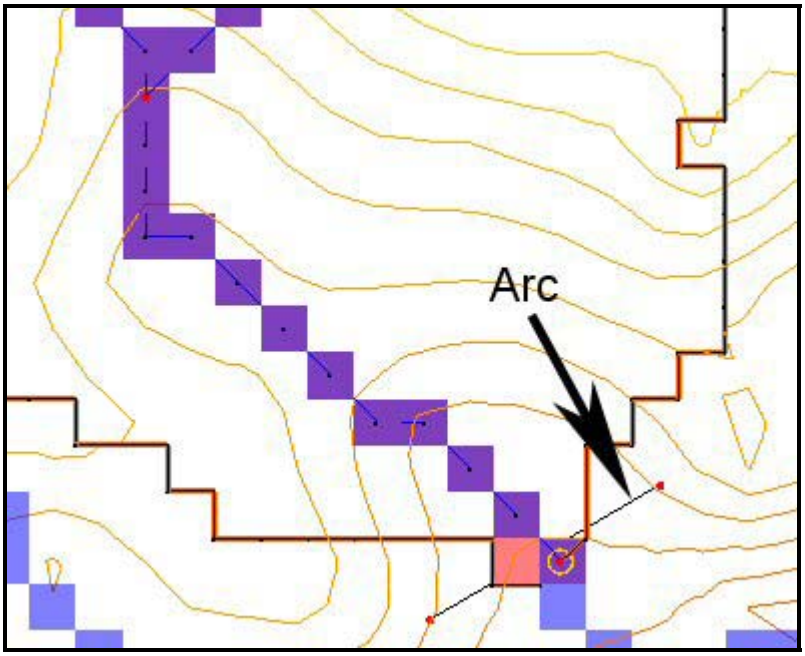


Figure 8-15: Feature arc location

2. Select the arc with the *Select Feature Arc* tool
3. Select ***DEM | Edit Elevations…***
4. Select the *Interpolate* button
5. Use the *Select Point* tool to edit the elevation point in the middle of the profile
6. Select *OK* when you see the error message about rerunning TOPAZ

The DEM cell represented by the selected profile point is highlighted in the main graphics window

7. Make sure that the selected profile point corresponds to the DEM cell contains the outlet
8. Drag the selected profile point down approximately to its original elevation, which is represented by the dotted red line in Figure 8-16

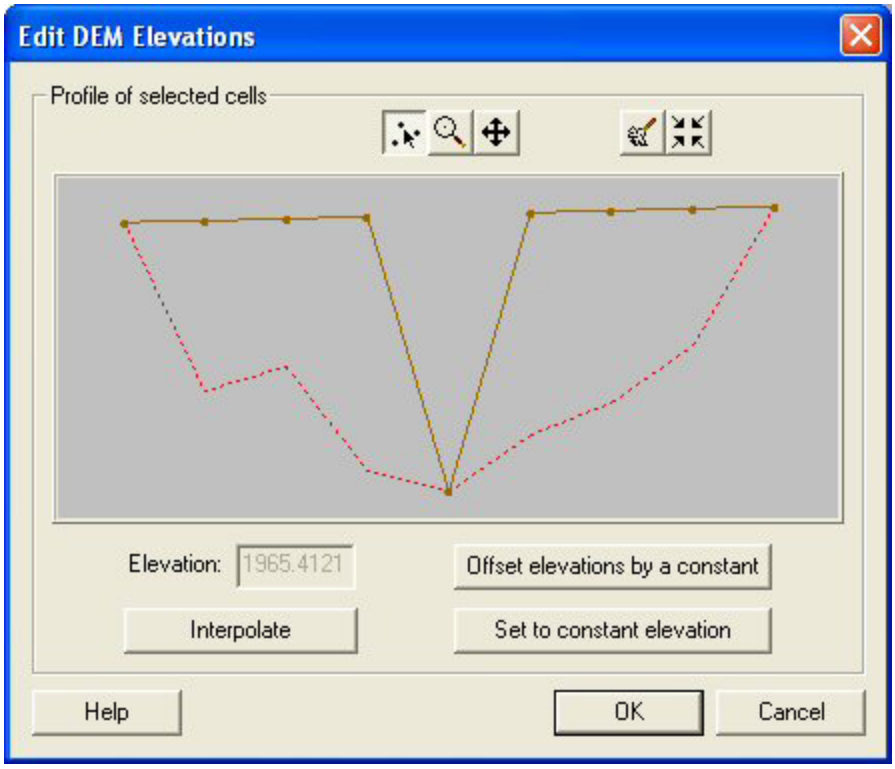


Figure 8-16: Editing DEM elevations

9. Select *OK* and notice the updated contours

We have effectively edited the DEM elevations to represent a small basin/reservoir created by the fill where the road crosses over the drainage point.

8.6.3 Run TOPAZ

1. Switch to the Drainage module
2. Select ***DEM | Compute TOPAZ Flow Data…***
3. Select *OK*
4. Select *OK*
5. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

8.6.4 Edit Flow Directions

Repeat the steps in Section 8.4.4 to correct the flow directions again

8.6.5 Delineate Basin

1. Select ***DEM | Delineate Basins Wizard***
2. If prompted, select *OK* to delete all existing feature data and recreate it using the new basin delineation
3. Select *OK*
4. Select *OK*

8.7 Computing a Storage Capacity Curve

A storage capacity curve can be computed based on DEM elevations for any outlet.

1. Use the *Select Nodes tool* to select the outlet point
2. Select ***DEM | Create Reservoir***
3. Enter **1973** m for the *Water Surface Elevation*
4. Select *OK*

The output storage capacity curve will be in English units of acre-ft vs. ft.

5. Select *OK* to close the plot of the storage capacity curve

8.8 Hydrograph Routing

An input hydrograph can be routed through any reservoir or basin on a DEM by using the storage capacity and elevation discharge curves that are defined in the Detention Basin calculator.

8.8.1 Open an Input Hydrograph

1. Switch to the Hydrologic Modeling module
2. Use the *Select Outlet* tool to select the outlet
3. Select ***Hydrographs | New Hydrographs | From File…***
4. Open *"hydro.txt"*
5. Leave the default settings because this is a tab delimited file. Select *Next*

6. In the *File preview* spreadsheet, map the first column type to ***Time*** and the second column type to ***Flow***
7. Select *Finish*
8. Use the *Select Hydrograph* tool to select the hydrograph
9. Select ***Display | Open Hydrograph Plot*** to view the input hydrograph
10. Close the hydrograph window by selecting the X in the upper right of the window

8.8.2 Elevation Discharge Curve

1. Select ***Calculators | Detention Basins…***
2. Select the *Define…* button
3. Click the *Define Discharges…*button
4. Select the *Add Outlet* button
5. Enter **3.14** ft^2 for *Outlet area*
6. Enter **6450.5** ft for *Outlet elevation*
7. Enter **6448.5** ft for *Base elevation*
8. Select *OK*
9. Select *OK*

8.8.3 Compute the Routed Output Hydrograph

1. Select the *Compute Outflow Hydrograph* button to view a plot of the input and routed hydrographs
2. Select *OK*
3. Select ***Display | Open Hydrograph Plot*** to view both the input hydrograph and the routed hydrograph

CHAPTER 9

Time of Concentration Calculations and Computing a Composite CN

Estimated Time: 30 minutes.

Travel times (time of concentration, lag time, and travel time along a routing reach) are critical to performing analysis with any of the hydrologic models. In this tutorial you will learn about the two different ways WMS can be used to compute time of concentration for a TR-55 simulation (lag times are computed in the same way):

- Runoff distances and slopes for each basin are automatically computed whenever you create watershed models from TINs or DEMS and compute basin data. These values can then be used in one of several available equations in WMS to compute lag time or time of concentration.

- If you want to have a little more control (and documentation) over the lag time or time of concentration, you will use a time computation coverage to define critical flow paths. Time computation coverages contain flow path arc(s) for each sub-basin. An equation to estimate travel time is assigned to each arc and the time of concentration (or lag time) is the sum of the travel times of all arcs within a basin. Lengths are taken from the length of the arc and slopes derived if a TIN or DEM are present.

In this tutorial you will compute the time of concentration for the two sub-basins and the travel time between outlet points in the watershed shown below.

You will use the TR-55 library of equations, but you could just as easily use one of the other pre-defined equations, or enter your own equation.

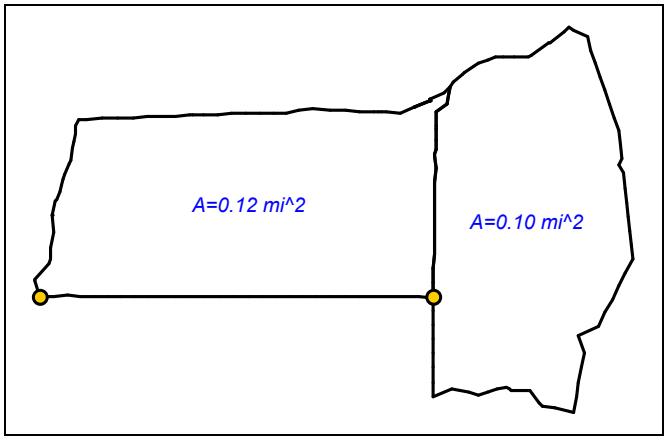


Figure 9-1 American Fork Watershed

Many hydrologic models, including TR-55, use a composite curve number to account for losses. A composite curve number is computed for each basin by overlaying land use and soil type polygons.

9.1 Reading a TIN File

You will first read in a TIN that has already been processed and used to delineate two sub-basins. The TIN serves the same purpose as a drainage coverage combined with a DEM.

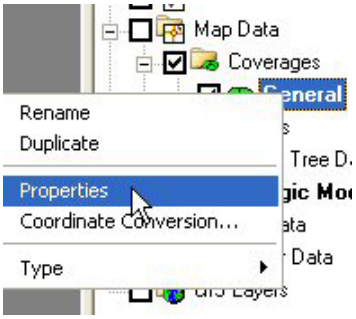
1. Switch to the Drainage Delineation module
2. Select ***File*** | ***Open…***
3. Open "*aftr55.tin"*
4. Select ***TIN*** | ***Compute Basin Data…***
5. Select the *Current Coordinates* button
6. Make sure that the Horizontal and Vertical units are set to *U.S. Survey Feet*
7. Select *OK*
8. Select *Square miles* for Basin Areas, and *Feet* for Distances
9. Select *OK*

10. Select ***Display | Display Options…***
11. Choose the *TIN* tab
12. Toggle off the check box for *Triangles*
13. Choose the *TIN Drainage* tab
14. Toggle on the box for displaying *Drainage Basin Boundaries*
15. Select *OK*

Your display should now show the basin boundaries and stream network.

9.2 Defining Flow Path Arcs

Flow paths can automatically be traced across a TIN (or DEM with flow directions computed) using the flowpath tool. You can take advantage of this functionality in *WMS* to set up your flow path segments.



1. Expand the *Map Data* folder in the Data Tree if necessary
2. Right-click on the *General* coverage in the *Coverages* folder of the Data Tree
3. Select *Properties…* from the pop-up menu
4. Set the Coverage type to *Time Computation*
5. Select *OK*
6. Select the *Create Feature Points* tool
7. Create feature points at the two locations marked by an *X* in the figure below. Be sure that there is one point inside of each basin boundary.

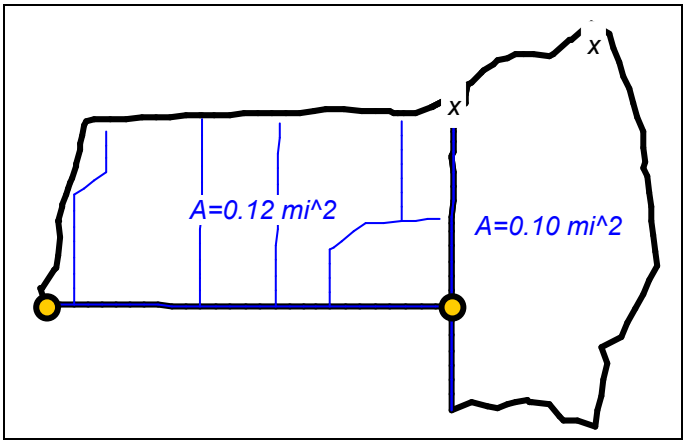


Figure 9-2: Starting Locations for Time of Concentration Arcs.

These points represent the furthest point from the outlet for their respective basins. Now feature arcs will be created from these points to the outlet by following the flow path.

1. Select the *Select Feature Point/Node* tool
2. Select both feature points just created (use the <SHIFT> key to multi-select)
3. Select ***Feature Objects | Node->Flow Arcs***
4. Choose the *Create multiple arcs* option
5. Select *OK*

The *Create multiple arcs* option will cause *WMS* to break the flow path arcs once they enter the stream defined on the TIN. The TR-55 method (others are similar as well) uses three different flow segments to compute time of concentration: sheet flow (up to 300 feet), shallow concentrated flow, and open channel flow. Since *WMS* will automatically break the arcs between overland and channel flow, two of three segments will already be defined. You will need to separate the sheet flow from the shallow concentrated flow before setting up the equations.

1. Select the *Select Feature Vertex* tool
2. The figure below identifies locations approximately 200-300 feet downstream from the beginning of the flow path arc. Select one of these vertices.
3. Select ***Feature Objects | Vertex<->Node***

4. Repeat for the other flow arc (alternatively you could multi-select the two vertices and convert them to nodes at the same time).

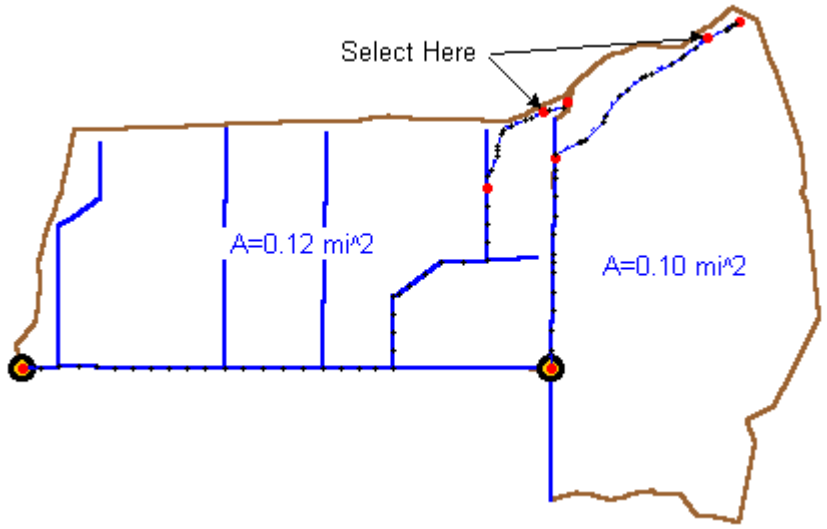


Figure 9-3: Locations to Break Sheet Flow Segments.

You should now have three arc segments for each basin. These arcs will be used to compute a time of concentration for the TR-55 analysis. However, we will also want to compute a travel time for the flow from the upper basin to the lower basin. This will require a flow path segment between the upstream outlet and the lower outlet. The flow path arcs from the lower basin already include flow for part of this distance, but there remains a segment for which no flow path arc exists.

1. Select ***Feature Objects | Streams->Flow Arcs***

This will create an arc that joins the outlet of the upper basin with the flow path segment already created for the lower basin. Where the flow path segments meet the arc from the lower basin is split into two. This will leave four arcs to define time of concentration for the lower basin (two will actually be in the stream). The arc just created, while in the lower basin, will not be used when computing time of concentration.

You have used the Node->Flow Arcs and Streams->Flow Arcs commands to automatically generate the flow path segments from flow on a TIN. Similarly, arcs could have been created from flow on a DEM with flow directions, or they could be created manually using the create arc tool . When defining arcs manually you must keep in mind that just like stream arcs in the drainage coverage, direction is important. Time computation arcs should be defined from downstream to upstream.

9.3 Assigning Equations to Time Computation Arcs

With the flow path segments defined you are now ready to assign equations that will be used to determine a travel time for the arc. Use the following figure as a guide while defining the equations.

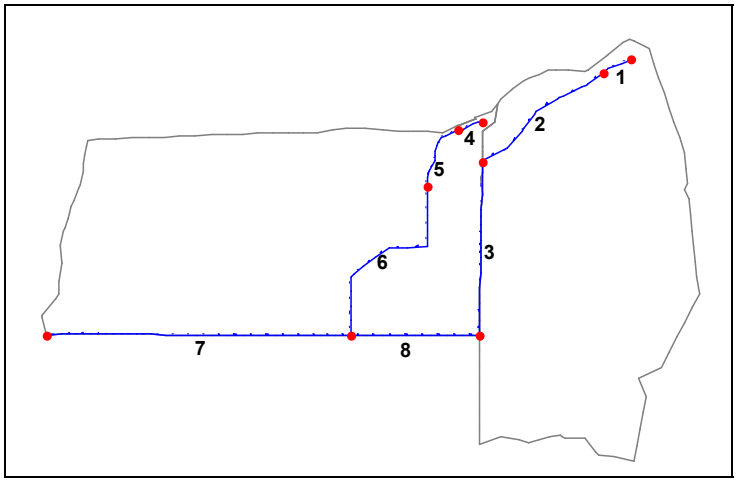


Figure 9-4: Time Computation Arcs.

1. Choose the *Select Feature Arc* tool
2. Double-click on the arc labeled *1*

By default the arc will be a TR-55 sheet flow equation arc, so all you need to do is define the overland Manning's roughness coefficient and the 2yr-24hr rainfall. Length and slope will already be defaulted from the selected arc.

1. Click on the *n Mannings* line in the *Variables* text window
2. Enter a value of **0.24** in the *Variable value* edit window
3. Click on the *rainfall* line in the *Variables* text window
4. Enter a value of **1.1**

You should notice in the Instructions/Results window that you are told what variables need to be defined before a travel time can be computed. Once you have entered all the necessary values, this same window reports the travel time for this arc. In this way, you can compute travel time for any arc segment no matter what the application is.

1. Select *OK*
2. Repeat the previous steps for the arc labeled 4. Use a Manning's roughness of **0.15** and a rainfall value of **1.1**

3. Select *OK*

You have now defined equations for the overland sheet flow segments in each basin and are ready to define the next two arc segments as shallow concentrated flow.

1. Double-click on the arc labeled *2*
2. Change the equation type to *TR-55 shallow conc eqn*
3. Click on the *Paved* line in the *Variables* text window
4. Enter **no** in the *Variable value* edit window
5. Select *OK*
6. Repeat for the arc labeled 5, using the same equation type. In this case set the *Paved* value to **yes**

The remaining arcs will be defined as open channel flow arcs.

1. Double-click on the arc labeled *3*
2. Change the arc type to *TR-55 Open channel eqn*
3. Click on the *Manning's n* line
4. Enter a value **0.016** in the *Variable value* edit window
5. Select the *hydraulic radius* line in the *Variables* window
6. Select the *Hydraulic Radius* button to open up the channel calculator so that the hydraulic radius can be computed from rough estimates
7. Change the *Channel type* to *Rectangular*
8. Enter a *Longitudinal slope* of **0.0015**
9. Enter a *Channel width* of **3.0**
10. Choose the *Enter depth* option
11. Enter a depth of **0.33** (an approximated depth since we do not know what the flow is at this point)
12. Select the *Calculate* button
13. Select *OK* for both dialogs

You have now defined the necessary parameters for computing travel time using the TR-55 open channel flow (Manning's) equation. Repeat for the arcs labeled 6, 7, and 8 using the following values:

Arc	Manning's N	Hydraulic Radius
6	0.016	0.35
7	0.016	0.41
8	0.016	0.38

If you wish you can continue to experiment with the channel calculator to compute the hydraulic radius rather than entering the given values.

You now have defined equations and variable values for each flow path segment. You can change these equations, variables, add new flow path segments, etc. in order to determine the best flow paths and most appropriate equations for each basin. In other words, this process is subjective and it may take a few iterations to get the best value.

9.4 Using the Time Computation Arcs to Compute Time of Concentration for a TR-55 Simulation

Before assigning time of concentrations to each basin you need to decide which model you want to use. For this tutorial you will be running *TR-55*, but the same time computation tools you learn in this tutorial could be used for any of the supported WMS models (such as in the *TR-20* basin data dialog, the *HEC-1* Unit Hydrograph method dialog, and the *Rational Method* dialog).

1. Select the *Hydrologic Modeling* module
2. Change the Model drop-down list to *TR-55*
3. Select ***TR-55* | *New Simulation***
4. Select ***TR-55* | *Run Simulation…***
5. Select the *Select Basin* tool while the *TR-55* dialog is still open
6. Select the upper basin
7. Enter a *Rainfall* value of **1.5**
8. Change the Rainfall distribution to *Type II*
9. Select the *Compute Tc – Map Data* button. You will see the three time computation arcs that are in the basin
10. You can create a detailed report as a text file if you want by selecting the *Export Data* or *Copy to Clipboard* buttons

11. Select *Done*

12. The sum of the travel times for these arcs will be used as the time of concentration for this basin

Note that you could bring up the time computation attributes dialog and change the equation or any of the equation variables by selecting the *Edit Arcs* button. Now you will set up the TR-55 basin data for the lower basin.

As a comparison you will now compute the time of concentration using the pre-computed runoff distance and slope parameters and one of the equations defined in WMS. To see the values that WMS has computed you will need to modify the display options.

1. Select the *Done* button to close the *TR-55* dialog

2. Select ***Display | Display Options…***

3. Choose the *Drainage Data* tab

4. Toggle on *Max flow distance* and *Max flow slope*

5. Select *OK*.

Notice that the longest flow path distance within a basin is displayed for each basin as well as the slope along that flow path. These are the variables that will be used in a separate equation to determine the time of concentration.

1. Double-click the upper basin icon to run TR-55 once again

2. Select the *Compute TC - Basin Data* button

In the Basin Time Computation dialog you can pick from one of several lag time or time of concentration equations that are pre-programmed in WMS (see *WMS Help*, topic: *Overview of Basin Data Equations*) for a complete description of these equations). You can also create your own equation from the computed variables, or parameters that you define yourself. In this tutorial we will use the Kerby equation for time of concentration, but you may wish to experiment on your own with some of the other equations or by making your own.

1. Select the *Compute Time of Concentration* option from the Computation type drop-down box

2. Choose the *Kerby method for overland flow* equation in the drop-down list of equations. Notice that all variables needed for this equation are defined and so the time of concentration is reported in the text window at the top of the dialog

3. Select *OK* to close the *Basin Time Computation* dialog

You should notice that the time of concentration is very nearly the same as before (.57 hours).

At this point you might be wondering why you would ever want to create a time computation coverage and go through the effort of defining travel time arcs and assign equations to each if the same value can be determined using the parameters already computed from a TIN or DEM?

1. Select the *Compute TC - Basin Data* button again
2. Select the *Kirpich method for overland flow on bare earth* equation
3. Select *OK*

You should now set the time of concentration back to the one originally computed by the flow arc method. To do this:

1. Select the *Compute Tc – Map Data* button
2. Select *Done*

The lesson here is the equations using Basin Data work great if you are confident the equation represents the same kind of land surface you are dealing with. If not, you are likely better off to define equations along separate flow path segments using the Map Data. Of course, in either case you can customize the equations to suit your own needs.

1. Select the lower basin from the graphics window
2. Enter a *Rainfall* value of **1.5**
3. Change the Rainfall distribution to *Type II*
4. Select the *Compute Tc – Map Data* button

This time you will notice that there are more than three arcs. This is because one of the flow path arcs (the one labeled 8 above) will be used to compute travel time between outlets but still is within the lower basin. Notice that WMS defaults its status to inactive. WMS will try to make the best choice of arcs that are used for time of concentration for a basin, but at times you may need to set the active/inactive status on your own.

1. Select *Done*

9.5 Using the Time Computation Arcs to Compute the Travel Time Between Outlet Points

Besides using time computation arcs for computing time of concentration or lag time for a basin, you can also use them to compute the routing travel time of a channel.

1. Choose the *Select Outlet* tool while the *TR-55* dialog is still open
2. Select the upstream outlet (the outlet to the upper basin)
3. Select the *Compute travel time…* button in the *TR-55* dialog
4. Make sure that both of the downstream arcs are active (The active status in the dialog is turned on for both)
5. Note the time of concentration for these arcs
6. Select *Done*
7. The time of concentration from the arcs should now appear in the *Travel time* box.
8. Select *Done* to close the TR-55 dialog

9.6 Computing a Composite Curve Number

In this part of the workshop, you will learn how to overlay land use and soil coverages on your delineated watershed in order to derive a curve number (CN).

1. Select the *Map* module
2. Select ***File | Open…***
3. Open *"luse.jpg"*

This file is an image of land use regions for an area. You can either digitize the polygons as demonstrated in the basic feature objects tutorial or read in a prepared file that has this already done for you. To open the map file:

4. Open "*luse.map"*

9.6.1 Land Use Table

Now you need to create a land use table with ID's and CN's for each type of land use on your map. To create the table, complete the following steps:

1. Choose the *Select Feature Polygon* tool
2. Double-click on the area labeled *11*

You can see from the WMS landuse ID section of the dialog box that the three land use ID's have already been added.

1. Make sure the *Display SCS CN's* check box is selected
2. In the *Land use properties* box, select the *Land name* line
3. In the *Land Name* edit box, type **Residential**
4. In the *Land use properties* box, select the *SCS Soil Type A CN* line
5. In the *Type A CN Value* edit box, type **61**
6. Repeat for Type B, C, and D, using values of **75**, **83**, and **87**
7. In the *WMS landuse ID* box, select Land ID ***13***
8. Enter the appropriate land names and Type CN's from the table below
9. Repeat this procedure for the other landuse ID's
10. Choose *Close* to exit the dialog (you don't want to choose apply or the last ID you are editing, 22, will be applied to the polygon you selected to open this dialog in the first place, 11)

Table 9-1: Landuses and CN's

ID	Name	Type A CN	Type B CN	Type C CN	Type D CN
11	Residential	61	75	83	87
13	Industrial	81	88	91	93
22	Orchards, Groves, Vineyards, Nursuries	36	60	73	79

9.6.2 Computing Composite Curve Numbers

In order to compute composite curve numbers, WMS needs to know which type of soil underlies each area of land. You will need to create a soil type coverage, create soil type polygons, and enter soil type data in the same manner as for land use. For this tutorial, this has already been done for you. You will need to read in this file:

1. Select ***File*** | ***Open…***
2. Open "*soils.map"*
3. Make sure the *Select Feature Polygon* tool is still the active tool

4. Select ***Feature Objects*** | ***Attributes…***
5. Select *Soil ID 0* from the WMS soil ID list

You will note that a soil with ID = 0 is automatically assigned to be Type A. Similarly, ID = 1 is B, 2 is C, and 3 is D.

6. Select *Close*
7. Select ***Display*** | ***Display Options…***
8. Toggle on *Color Fill Polygons*
9. Switch to the *General* tab
10. Toggle on *Soil Type Legend*
11. Select *OK*

Now it is easy to see which soil types are assigned to which polygons.

12. Select ***Display*** | ***Display Options…***
13. Toggle off *Color Fill Polygons*
14. Select *OK*
15. Switch to the *Hydrologic Modeling* module
16. Select ***Calculators*** | ***Compute GIS Attributes…***
17. Make sure the *SCS Curve Numbers* option is selected in the *Computation* section of the dialog
18. Specify to *Use a Soil type coverage for determining soil type*
19. Select your soil type coverage name ("*Soil Type*") from the *Soil type coverage name:* drop down list
20. Specify to *Use a Land use coverage for determining land use*
21. Select your land use coverage name ("*Land Use*") from the *Land use coverage name:* drop down list

9.6.3 Importing Land Use and Soil Type Tables

You may have your land use and soil type tables stored in data files. Instead of manually assigning the data as you have here, you would read these tables in from this dialog using the *Import* button.

Whether you have manually created tables or read them in from files, you should see the land use ID's, CN's for each soil type, and land use descriptions in the window of the Mapping section.

1. Select *OK* to compute the composite CN's

The composite CN for each basin should now be displayed at the centroid of each basin.

9.7 More TR-55

While you were entering the data for each of the two basins you may have noticed that instructions are given in the TR-55 data window to let you know what must be entered before a peak Q can be determined. Once you enter all of the data the peak Q is computed and displayed in this same window. You can also get help for anything listed in this window.

1. Select the *Select Basin* tool with the *TR-55* dialog still open
2. Select the upper basin
3. Select the line in the TR-55 data window where the peak Q is displayed

Notice that the TR-55 equation for computing peak flow is displayed in the help window. You can get help for missing data as well.

1. Change the Time of concentration to **0.0** and click somewhere else to update the dialog
2. Select the line in the TR-55 data window that instructs you to define a time of concentration

Now you are told the units for time of concentration as well as the available options for determining it in the TR-55 help window.

1. Reenter the time of concentration by typing in the appropriate value, or if you can't remember use the *Compute Tc – Map Data* button again
2. Select the *Compute Hydrograph(s)* button
3. Select *Done* to close the *TR-55* dialog

4. Choose the *Select Hydrograph* tool

5. Double-click on the hydrograph icon that is displayed by the upper basin to view the hydrograph in a separate window

9.8 Conclusions

This completes the tutorial on using the time computation coverage to compute time of concentration and travel times and the land use and soil coverages to compute a composite CN value. In the process you have also learned about the TR-55 interface.

CHAPTER 10

HEC-1 Interface

WMS has a graphical interface to HEC-1. Geometric attributes such as areas, lengths, and slopes are computed automatically from the digital watershed. Parameters such as loss rates, base flow, unit hydrograph method, and routing data are entered through a series of interactive dialog boxes. Once the parameters needed to define an HEC-1 model have been entered, an input file with the proper format for HEC-1 can be written automatically. Since only parts of the HEC-1 input file are defined in this tutorial, you are encouraged to explore the different available options of each dialog, being sure to select the given method and values before exiting the dialog.

The US Army Corps of Engineers now supports HMS rather than HEC-1, but the hydrologic calculations for the options within HEC-1 have not changed. Results between the two models will be identical.

10.1 Objectives

As a review you will delineate a watershed from a DEM. You will then develop a simple, single basin model using the delineated watershed to derive many of the parameters. Land use and soil shape files (downloaded from the internet) will be used to develop a SCS curve number (CN) value. After establishing the initial HEC-1 model other variations will be developed, including defining multiple basins with reach routing and including a reservoir with storage routing.

10.2 Delineating the Watershed

Since the land use, soil type, and DEM data for our watershed all originate in the Geographic coordinate system, we will begin by opening them together and converting them to UTM coordinates. The land use and soil type data were downloaded from the EPA website. The DEM data used for this watershed were previously downloaded from the National Elevation Dataset as was demonstrated in the DEM basics tutorial.

1. Select ***File*** | ***Open…***
2. Select *NED GRIDFLOAT Header (*.hdr)* from the Files of type list of file filters
3. Find and open "*67845267.HDR*"
4. Select *OK*
5. When prompted if you want to convert the current coordinates select *No*

10.2.1 Create Land Use and Soil Coverages

1. If it is not expanded yet, expand the *Map Data* folder in the Data Tree window by clicking on the plus mark next to the folder icon
2. Right click on the *Coverages* folder in the Data Tree
3. Select *New Coverage*
4. Change the Coverage type to *Land Use*
5. Select *OK*
6. Create a new coverage once again and set its Coverage type to *Soil Type*

10.2.2 Open the Soils Data

1. Make sure the *Soil Type* coverage is active in the Data Tree
2. Switch to the GIS module
3. Select ***Data*** | ***Add Shapefile Data…***
4. Open "*statsgo.shp*"
5. Expand the *GIS Layers* folder in the Data Tree if necessary

6. Right click on *statsgo.shp* layer in the Data Tree
7. Select *Open Attribute Table*

Notice that the table has three fields named AREA, PERIMETER, and MUID

8. Select *OK*

10.2.3 Join Soils Database File Table to Shapefile Table

1. Right click on *statsgo.shp* in the Data Tree
2. Select *Join Table to Layer…*
3. Open "*statsgoc.dbf*"
4. Ensure that Shapefile Join Field and Table Join Field are both set to *MUID*
5. Change the Table Data Field to *HYDGRP*
6. Select *OK*
7. Right click on *statsgo.shp* in the Data Tree
8. Select *Open Attribute Table*

Notice that the HYDGRP field is now a part of the shapefile.

9. Select *OK*

10.2.4 Convert Soil Shapefile Data to Feature Objects

1. Choose the *Select Shapes* tool
2. Draw a selection box around the DEM extents
3. Select ***Mapping | Shapes -> Feature Objects***
4. Select *Next*

This window shows all of the attribute fields in the soils shape file. Because this file was derived from a standard NRCS statsgo file you will notice that the hydrologic soil groups field is named HYDGRP and so WMS will automatically map this to be the soil type. If the attribute field were named anything other than HYDGRP then you would have to manually map it using the drop down list in the spread sheet.

5. Make sure the *HYDGRP* field is mapped to the *SCS soil type* attribute
6. Select *Next*
7. Select *Finish*
8. Clear the selected polygons by single-clicking somewhere beyond the extents of the shapefile polygons
9. Hide the *statsgo.shp* file by toggling off its check box in the Data Tree

10.2.5 Open the Land Use Data

1. Switch to the Map module
2. Select the *Land Use* coverage in the Data Tree to designate it as the active coverage
3. Switch to the GIS module
4. Select ***Data*** | ***Add Shapefile Data…***
5. Open "*l_richut.shp*"
6. Choose the *Select Shapes* tool
7. Draw a selection box around the DEM extents
8. Select ***Mapping*** | ***Shapes -> Feature Objects***
9. Select *Next*
10. Make sure the *LUCODE* field is mapped to the *Land use* attribute
11. Select *Next*
12. Select *Finish*
13. Hide the *l_richut.shp* file by toggling off its check box in the Data Tree

10.2.6 Convert/Set the Coordinate System of the Data

1. Select ***Edit*** | ***Coordinate Conversion…***
2. Toggle the *Edit project coordinate system* option on

3. In the *Convert From…* tab define the Horizontal System to be *Geographic NAD 83 (US)*

4. Set the Vertical Units to *Meters*

5. In the *Convert To…* tab define the Horizontal System to be *UTM NAD 83 (US)*. Zone 12 will automatically be set.

6. Make sure the Horizontal and Vertical Units are *Meters*

7. Select *OK* to convert the data

8. Switch to the Map module

9. Since we won't be using them until later, hide the *Land Use* and *Soil Type* coverages by toggling off their check boxes in the Data Tree

10. Select the *Drainage* coverage from the Data Tree to make sure it is the active coverage

10.2.7 Delineate the Watershed

1. Select the *Drainage* module

2. Select the *Frame* macro

3. Select ***DEM | Compute Topaz Flow Data…***

4. Select *OK*

5. Select *OK* in the Units dialog

6. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

7. Select the *Zoom* tool

8. Zoom in by dragging a box as illustrated in Figure 10-1

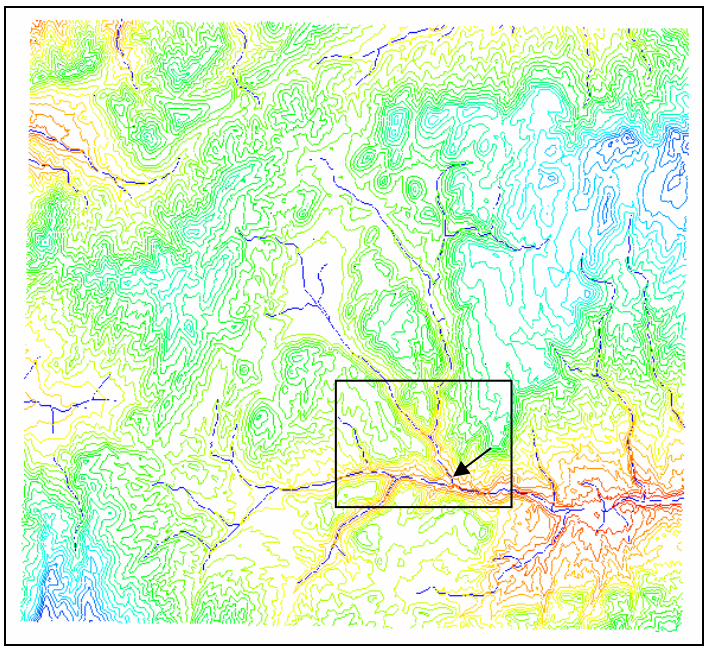


Figure 10-1: Zoom in on the area bounded by the rectangle above

9. Select the *Create outlet point* tool

10. Create a new outlet point where the tributary you just zoomed in on separates from the main stream as illustrated by the arrow in Figure 10-1. Make certain that the outlet point is on the tributary and not part of the main stream. Also, the outlet needs to be inside one of the flow accumulation (blue) cells. WMS will move the outlet to the nearest flow accumulation cell if you do not click right in one of the flow accumulations cells.

11. Select the *Frame* macro

12. Select ***DEM | Delineate Basins Wizard***

13. Select *OK*

14. Select *OK*

You have now completed the delineation of a single watershed. In order to make the view clearer for defining the hydrologic model you can turn off many of the DEM and other display options.

15. Select the *Display Options* macro

16. On the DEM tab toggle off the display for *Watershed*, *Stream*, *Flow Accumulation*, and *DEM Contours*

17. On the Map tab toggle off the display of *Vertices*

18. Select *OK*

10.3 Single Basin Analysis

The first simulation will be defined for a single basin. You will need to enter the global, or Job Control parameters as well as the rainfall event, loss method, and unit hydrograph method.

10.3.1 Setting Up the Job Control

Most of the parameters required for a HEC-1 model are defined for basins, outlets, and reaches. However, there are many "global" parameters that control the overall simulation and are not specific to any basin or reach in the model. These parameters are defined in the WMS interface using the Job Control dialog.

1. Select the *Hydrologic Modeling* module

2. HEC-1 should be the default model, but if it is not select it from the drop down list of models found in the Edit Window

3. Select ***HEC-1*** | ***Job Control…***

4. The first three lines are for comments/identification at the top of the HEC-1 input file. The first line already has information indicating that the input file is generated by WMS (you can change this if you want). Enter **Clear Creek Tributary Watershed** for the second ID line. Enter your name and current date in the third line.

5. Leave the Day, Month, and Field years alone

HEC-1 allows you to enter a date, but almost always you are simulating some kind of hypothetical or design storm and not an actual storm. If you change the simulation date you will need to be careful to make sure the storm date is in synch, but if you leave it alone then there will not be a problem.

6. Enter **5** (minutes) for the Computation time interval, and **300** for the Number of hydrograph ordinates. Leave the starting time at 0.

A HEC-1 simulation will run for a length of time equal to the time step multiplied by the number of ordinates. If you are simulating a 24-hr storm but only run the simulation for 12 hours you will not capture the full hydrograph. Conversely if you run a 24-hr simulation for 96-hrs you are probably going to have a lot of runoff ordinates equal to 0 at the end. In this case we are running

the simulation for 1500 minutes (slightly more than 24 hours) with an ordinate on the hydrograph being computed for every 5 minutes.

7. Set the computation units to *English* (this should be the default)

Setting the computation units DOES NOT cause any units conversion to take place. You are simply telling HEC-1 that you will provide input units in English units (sq. miles for area, inches for rain, feet/miles for length) and expect results of computation to be in English units (cfs). If you specify Metric then you must insure that input units are metric (sq. kilometers, mm for rain, meters/kilometers for lengh) and results will be in metric (cms).

8. Select *OK*

For now we will leave the other Job Control settings at their default values.

10.3.2 Setting up the Basin Data Parameters

In the first simulation you will treat the entire watershed as a single basin.

1. Select the *Select Basin* tool
2. Double-click on the brown basin icon labeled 1B. Double-clicking on a basin or outlet icon always brings up the parameter editor dialog for the current model (in this case HEC-1)
3. Select the *Basin Data* button
4. Notice that the area has been calculated (in this case in sq. miles because we are performing calculations in English units)
5. Change the name to **CCTrib**. HEC-1 will only use the first SIX characters so do not use names longer than six characters for basins or outlets.
6. Select *OK*
7. Select the *Precipitation* button
8. Select the *Basin Average* option
9. Enter **1.8** (inches) for the Average precipitation depth
10. Select the *Define Series* button

In order to simulate a rainfall event you must enter both a rainfall depth and a temporal distribution. The SCS uses standard time distributions for different areas of the U.S. These series have been created and stored in a file that you

can import. You could also define your own series according to an actual storm, or a design storm from a regulating agency.

1. Select the *Import* button
2. Find and open the file named "*scstabs.xys*". You may need to change the Files of type filter to *All Files (*.*)* in order to see the file.
3. In the Selected Curve drop down list select the *typeII-24hour* curve
4. Select *OK*
5. Select *OK*
6. Select the *Loss Method* button
7. Enter a Curve Number (CRVNBR field) of **70**. We will compute a CN value from actual land use and soil files later.
8. Select *OK*
9. Select the *Unit Hydrograph Method* button
10. Make sure the *SCS dimensionless* option is chosen (it is the default)
11. Select the *Compute Parameters - Basin Data* button
12. Set the Computation Type to *Compute Lag Time* (the default)
13. Set the Method drop down list to *SCS Method* (near the bottom of the list)
14. Select *OK* to update the computed lag time for the SCS dimensionless method
15. Select *OK*
16. Select *Done*

You now have all of the parameters set to run a single basin analysis.

10.3.3 Running HEC-1

Whenever you run a HEC-1 simulation WMS will first save a standard HEC-1 input file. You will also be prompted for the name of an output file and a solutions file. The output file is the standard text output file generated by HEC-1 and the solution file is a plot file that contains the hydrographs formatted in a way that makes it easy for WMS to read and plot (it is actually the HEC-1 TAPE22 file).

1. Select ***HEC-1 | Run Simulation…***

2. Change the text in the Prefix for all files field to be **CCTrib**

3. Select the *Update All Prefixes* button

4. Select *OK*

5. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

The solutions will automatically be read in and you should see a small hydrograph plot up and to the right of the Basin icon (now labeled as CCTrib).

1. Double-click on the hydrograph icon.

A plot window will appear with the hydrograph. You will see that the hydrograph suddenly stops at 1500 minutes (the duration of the simulation as established in the Job Control dialog), but the simulation obviously has not run to completion.

1. Close the plot window by selecting the X in the upper right corner of the window

2. Select ***Hydrographs | Delete All***

3. Select ***HEC-1 | Job Control…***

4. Set the Number of hydrograph ordinates to be **400**

5. Select *OK*

6. Select ***HEC-1 | Run Simulation…***

7. Select *OK* (you can let it overwrite the other files)

8. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

9. Double-click on the hydrograph icon

You now have a completed HEC-1 simulation for a single basin and the resulting hydrograph should look something like the solution shown in Figure 10-2.

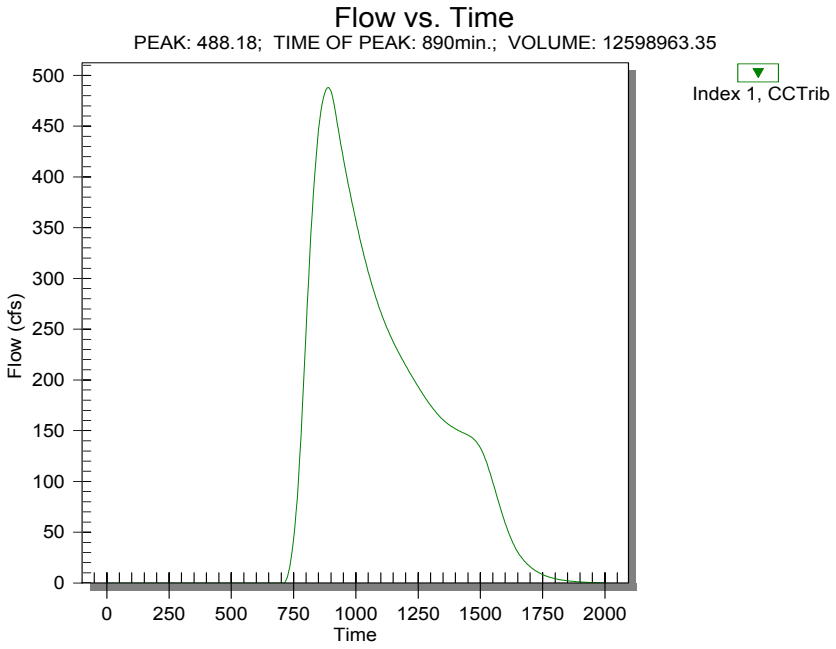


Figure 10-2: Solution hydrograph for HEC-1 simulation

1. Close the hydrograph window by selecting the X in the upper right of the window

10.4 Computing the CN Using Land Use and Soils Data

In the initial simulation you just estimated a CN, but with access to the internet it is simple to compute a composite CN based on digital land use and soils files. This was demonstrated in more detail in the Advanced Feature Objects tutorial, but you will go through the steps here as a review.

10.4.1 Computing a Composite CN

In addition to the digital land use and soils file that overlap the watershed you must have a table defined that identifies CN values for each of the four different hydrologic soil groups (A, B, C, D). This is described in detail at the gsda website (http://emrl.byu.edu/gsda), and in the advanced feature objects tutorial. For this tutorial you will read in an existing file (you can examine it in a text editor if you wish) and compute the CN numbers.

1. Select the *Hydrologic Modeling* module
2. Select ***Calculators*** | ***Compute GIS Attributes…***
3. Select the *Import* button to load the mapping table

4. Select *OK* to overwrite the current definition

5. Find and open the file named "*scsland.tbl"*

6. Select *OK* to compute the CN from the land use and soils layers

You should find that CN computed from the land use and soils digital data is about 72 or 73. While there is still some "judgment" required in setting up the mapping table, there is a lot more justification for this value than the one previously estimated.

10.4.2 Run HEC-1

You can now run another simulation to compare the results with the modified CN value.

1. Select ***HEC-1 | Run Simulation…***

2. Select *OK* (it is fine to overwrite the existing files, but you can change the file names if you want)

3. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

4. Double-click on the hydrograph icon to plot both the old and the new hydrograph in a plot window

With the increased CN value you should see that the resulting hydrograph peaks higher (more runoff). The peak should be about 600 cfs rather than the 500 cfs that was generated with a CN value of 70.

1. Close the hydrograph window by selecting the X in the upper right corner of the window.

2. Select ***Hydrographs | Delete All***

10.5 Adding Sub-basins and Routing

You will now subdivide the watershed into two upper basins and one lower basin and define routing for the reaches that connect the upper basins to the watershed outlet.

10.5.1 Delineating the sub-basin

1. Select the *Drainage* module

2. Select the *Zoom* tool

3. Create a zoom box around the region identified by a box in Figure 10-3

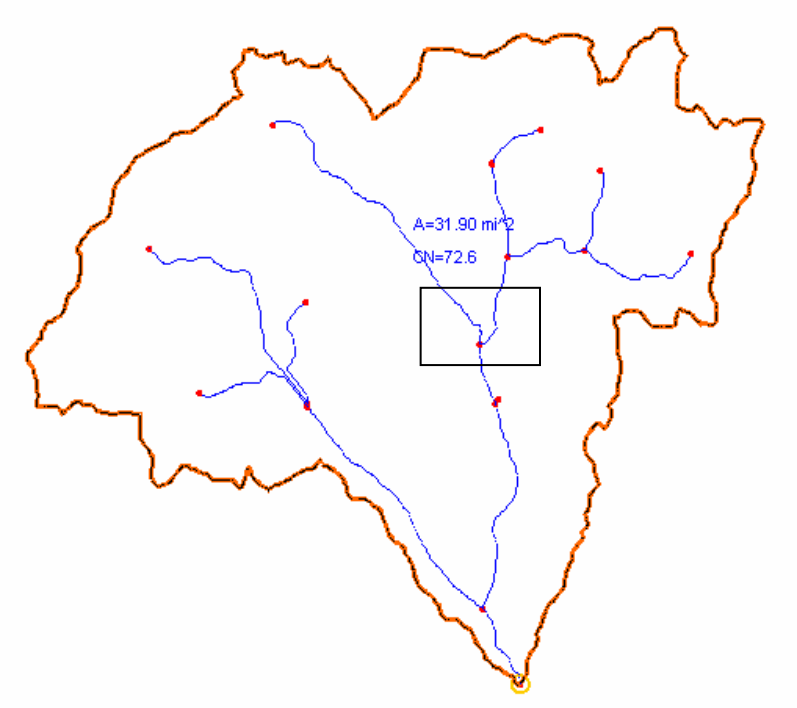
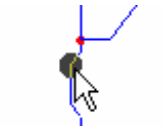


Figure 10-3: Zoom in on the area indicated by the rectangle.

1. Select ***Display | Display Options…***

2. On the Map tab toggle on *Vertices*

3. Select *OK*



4. Select the *Select Feature Vertex* tool

5. Select the vertex that is just below the main branching point you just zoomed in around

6. Select ***DEM | Node <-> Outlet***

You create the outlet point just below the branch in order to have a single upstream basin. If you wanted a separate basin for each upstream branch you could define the branching node to be an outlet. WMS will automatically assume that you want separate basins for each branch, so we have created a node just downstream of the branch and defined it as the outlet for the upper basin.

1. Select the *Frame* macro
2. Select the *Zoom* tool
3. Create a zoom box around the region identified by a box in Figure 10-4

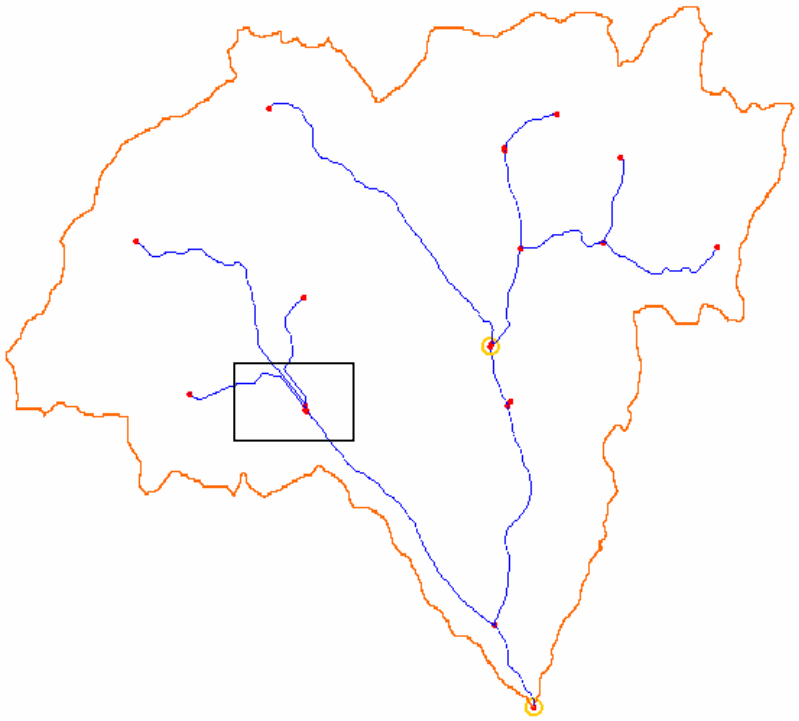
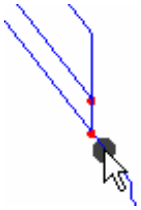


Figure 10-4: Zoom in on the area indicated by the rectangle

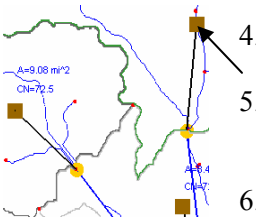


1. Select the *Select Feature Vertex* tool
2. Select the vertex that is just below the feature node where the streams branch
3. Select ***DEM | Node <-> Outlet***
4. Select the *Frame* macro
5. Select ***DEM | Delineate Basins Wizard***
6. Select *OK* to delete and recreate feature data
7. Select *OK*
8. Select *OK*

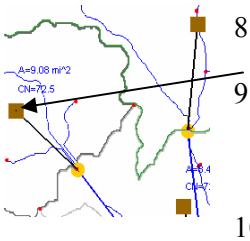
10.5.2 Updating the Basin Parameters

You will have to recompute the CN values and define precipitation and lag time for the basins.

1. Select the *Hydrologic Modeling* module
2. Select ***Calculators*** | ***Compute GIS Attributes…***
3. Select *OK* and the CN values will be updated for all basins (they are actually very similar in this case because of the dominant soil polygon that covers the watershed)



4. Select the *Select Basin* tool
5. Double-click on the upper right basin icon to bring up the *Edit HEC-1 Parameters* dialog
6. Select the *Basin Data* button
7. Change the Name to **Right**



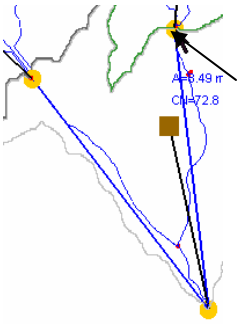
8. Select *OK*
9. Move the Edit HEC-1 Parameters dialog out of the way, if necessary, and click on the upper left basin icon to edit parameters for the upper left basin
10. Select the *Basin Data* button
11. Change the Name to **Left**
12. Select *OK*
13. Move the Edit HEC-1 Parameters dialog out of the way, if necessary, and click on the lower basin icon to edit parameters for the lower basin
14. Select the *Basin Data* button
15. Change the Name to **CCTrib**
16. Select *OK*
17. Select *Done*
18. Select ***Edit*** | ***Select All*** to select all basins
19. Select ***HEC-1*** | ***Edit Parameters…*** to edit parameters for all basins at once

20. Select the *Precipitation* button
21. Select the *Basin Average* option
22. Set the Average Precipitation to be **1.8**
23. Select the *Define Series* button
24. Choose the *typeII-24hour* curve in the Selected Curve drop down list
25. Select *OK*
26. Select *OK*
27. Select the *Unit Hydrograph Method* button
28. Make sure the *SCS dimensionless* option is chosen (it is the default)
29. Select the *Compute Parameters – Basin Data* button
30. Select *CCTrib* in the Basin window so that it is highlighted
31. Select the Method to be *SCS Method* (near the bottom of the list)
32. Select *Left* in the Basin window so that it is highlighted
33. Select the Method to be *SCS Method* (near the bottom of the list)
34. Select *Right* in the Basin window so that it is highlighted
35. Select the Method to be *SCS Method* (near the bottom of the list)
36. Select *OK*
37. Select *OK*
38. Select *Done*

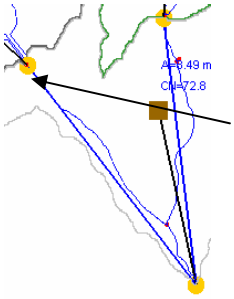
10.5.3 Setting up the Routing Parameters

If you were to run HEC-1 now (you can if you want) you would see that the hydrographs from the upper basins would be combined with the lower basin hydrograph at the watershed outlet without any lag or attenuation because you haven't yet set the routing parameters. You will now define a routing method, which will instruct HEC-1 to compute lag and attenuation on the upper basin hydrographs before adding them to the lower hydrograph.

Routing for a reach is always defined at the upstream outlet of the reach in WMS.



1. Select the *Select Outlet* tool
2. Double-click on the outlet (the yellow circle icon) of the upper right basin
3. Select the *Routing Data* button
4. Select the *Muskingum-Cunge* method for routing
5. Set the width (WD) field to be **5** (five feet wide)
6. Set the side slope value (Z) to be **1** (1:1 side slope)
7. Set the Manning's roughness (N) to be **0.05** (this is kind of rough but we want to exaggerate the routing effects for this tutorial)



8. Select *OK*
9. Select *Done*
10. Double-click on the outlet of the upper left basin
11. Select the *Routing Data* button
12. Select the *Muskingum-Cunge* method for routing
13. Set the width (WD) field to be **5**
14. Set the side slope value (Z) to be **1**
15. Set the Manning's roughness (N) to be **0.05**
16. Select *OK*
17. Select *Done*

10.5.4 Running HEC-1

You now have everything defined to run a three basin HEC-1 analysis that includes routing the upper basins through the reaches connecting them to the watershed outlet.

1. Select ***HEC-1 | Run Simulation…***
2. Change the name in the Prefix for all files field to **Routing**
3. Select the *Update All Prefixes* button

4. Select *OK*

5. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

6. While holding the <SHIFT> key down select all of the hydrograph icons, double-clicking on the last one so that all hydrographs are drawn in the same plot window

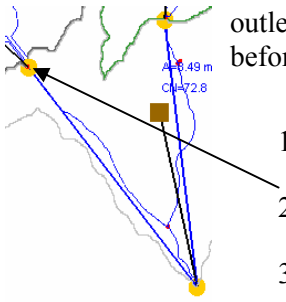
7. Close the plot window by selecting the X in the upper right corner

10.6 Modeling a Reservoir in HEC-1

There is an existing small reservoir at the outlet of the upper left basin. It has a storage capacity of 1000 ac-ft at the spillway level and 1540 ac-ft at the dam crest.

10.6.1 Defining a Reservoir in Combination with Routing

One of the routing methods available in HEC-1 is Storage routing, which can be used to define reservoir routing. However, in this case we are already using Muskingum-Cunge routing to move the hydrograph through the reach connecting the upper left basin to the watershed outlet so we must define the outlet as a reservoir so that we can route the hydrograph through the reservoir before routing it downstream.



1. Select the *Select Outlet* tool

2. Select the outlet of the upper left basin

3. Select ***Tree*** | ***Add Reservoir***

10.6.2 Setting up the Reservoir Routing Parameters

In order to define reservoir routing with HEC-1 you must define elevation vs. storage (storage capacity curve) and elevation vs. discharge rating curves. You can enter values directly, or enter hydraulic structures and compute the values, but in this tutorial you will enter the values directly. You will use the same elevation values for both curves (this is common, but not a requirement in HEC-1).

For this example we want to have no outflow until the elevation in the reservoir reaches the spillway. Since HEC-1 linear interpolates between consecutive points on the elevation-discharge and elevation-volume curves we will "trick" it by entering two points on the curves at essentially the same elevation

(6821.99 and 6822) with the first having no outflow and the second having the discharge over the spillway (640 cfs) as defined for this dam.

1. Double-click on the reservoir outlet point (it is now represented as a triangle since you have defined a reservoir at this location)

2. Select the *Reservoir Data* button

3. Change the Reservoir name to **Tcreek**

4. Set the Type of storage routing to *Reservoir*

5. Select the *Define* button to the right of the reservoir option

6. On the right side of this dialog you will define the Volume or storage capacity data. Choose the *Known Volume* option.

7. Toggle on the check boxes for *SV* (Volumes) and *SE* (Elevations)

8. Select the *Define* button to the right of the *SV* option

You will define separate XY series for Volumes, Elevations, and Discharges using the WMS XY Series editor.

1. Select *New*

2. Change the name of the new curve to **Volume**

3. In the first seven edit fields enter the values **0**, **200**, **410**, **650**, **1000**, **1000**, **1540** (acre-ft of volume)

4. Select the 8th through 20th edit fields and select the <DELETE> key so that the values are blank rather than zero. You can select them all at once (the way you do in a spreadsheet since this dialog behaves like a spreadsheet) by clicking in the top and while holding the mouse button down dragging to the last, or you can select one at a time.

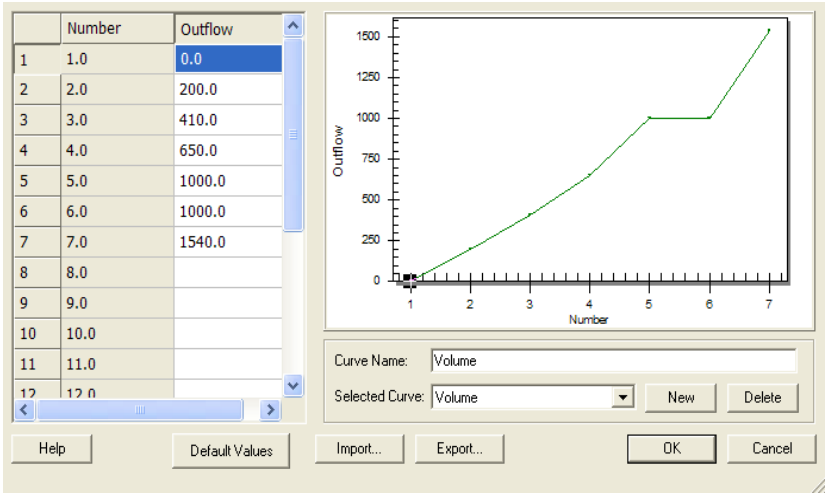


Figure 10-5: The XY Series editor for inputting volumes.

1. Select *OK*
2. Select the *Define* button to the right of the *SE* option
3. Select *New*
4. Change the name of the new curve to **Elevation**
5. In the first seven entry fields enter the following values: **6803**, **6808**, **6813**, **6818**, **6821.99**, **6822**, **6825** (feet of elevation)
6. Set the 8th through 20th fields blank instead of zero as with the volume series
7. Select *OK*
8. On the left side of this dialog you will define the Outflow or elevation - discharge data. Choose the *Known Outflow* option.
9. Toggle on the check boxes for *SQ* (Discharges) and *SE* (Elevations)
10. Select the *Define* button to the right of the *SQ* option
11. Select *New*
12. Change the name of the new curve to **Discharge**
13. In the first seven entry fields enter the following values: **0**, **0**, **0**, **0**, **640**, **640**, **7000** (cubic feet per second of flow). There is no outflow until the water reaches the spillway.
14. Set the 8th through 20th fields blank instead of zero as with the volume series

15. Select *OK*

16. Select the *Define* button to the right of the *SE* option

This time rather than creating a new curve you will select the elevation curve previously defined for the storage capacity curve.

1. Select the *Elevation* curve from the Selected Curve drop down list

2. Select *OK*

If you would like you may plot either the elevation-discharge or the elevation-volume curves by selecting the Plot SQ-SE or Plot SV-SE buttons. This will bring the curve into a plot window that you can export, print, or control the same way you can a hydrograph or any other plot in a plot window.

1. Select *OK*

The last thing you need to input to define reservoir routing is the initial conditions of the reservoir. The initial condition can be defined as an elevation, a discharge, or a volume (with the data you just entered HEC-1 can determine the initial condition of the other two based on the one you enter). For this example we will set the initial condition to an elevation four feet below the top of the spillway (the spillway corresponds to elevation 6822).

1. Under the Initial Condition Type select the *ELEV* option

2. Set the RSVRIC (reservoir initial condition) to be **6818**

3. Select *OK*

4. Select *Done*

10.6.3 Running HEC-1

You are now ready to save and run the HEC-1 file with the defined reservoir.

1. Select ***HEC-1 | Run Simulation…***

2. Change the name in the Prefix for all files field to **Reservoir**

3. Select the *Update all prefixes* button

4. Select *OK*

5. Select *Close* once HEC-1 finishes running (you may have to wait a few seconds to a minute or so)

6. After HEC-1 runs you can open any (or multiple within the same plot window by holding down the <SHIFT> key to multi-select) of the hydrographs by double-clicking on the corresponding icon.

7. Close all plot windows before moving on

10.7 Reviewing Output

It should be emphasized here that while WMS makes it easy to set up a HEC-1 model and compute a result, it is not a substitute for understanding the basic theory and equations used in HEC-1. You are encouraged to read the HEC-1 manual found in the documents directory distributed with WMS and other texts on hydrologic modeling. You are also encouraged to review the HEC-1 output file that is generated with each simulation in order to glean more understanding about how your model is working.

1. Select ***File*** | ***Edit File…***

2. Find and open the file named "*reservoir.out*"

3. Select OK to open the file with *Notepad*

4. Scroll through this file and examine what information HEC-1 saves to the output file. If you have errors running HEC-1 simulations you may often find the answer to the problem within the *.out output file.

10.8 Conclusion

This concludes the tutorial on defining HEC-1 files and displaying hydrographs. The concepts learned include the following:

- Entering job control parameters
- Defining basin parameters such as loss rates, precipitation, and hydrograph methodology for a single watershed or a multiple basin analysis
- Defining routing parameters
- Routing a hydrograph through a reservoir
- Saving HEC-1 input files
- Reading hydrograph results

CHAPTER 11

Rational Method Interface

The Rational Method is one of the simplest and best known methods routinely applied in urban hydrology. Peak flows are computed from the simple equation:

$$Q = kCiA$$

where:

- Q - Peak flow
- k - conversion factor
- C - Runoff coefficient
- i - Rainfall intensity
- A - Area

In this tutorial you will learn how to solve problems using a digital terrain model and the Rational Method.

11.1 Reading in Terrain Data

The terrain model used in this tutorial is a TIN for a small portion of a city. The elevation data was obtained by digitizing a contour map.

1. Select ***File*** | ***Open…***

2. Find and open "*afrational.tin*"
3. Select the *Drainage* module
4. Select ***TIN*** | ***Compute Basin Data***
5. The Model Units should be feet. Set the Parameter Units to be Acres for *Basin Areas* and Feet for *Distances*.
6. Select *OK*

11.2 Running a Rational Method Simulation

The areas computed from the TIN can now be used in setting up a Rational Method simulation of the urban development. Each of the outlet points represents an inlet to a storm drain.

1. Select the *Hydrologic Modeling* module
2. Select *Rational* to be the current model from the drop down list of models in the Edit Window
3. Double-click the basin icon for the basin labeled "*Upper*" in Figure 11-1

The *Rational Method* dialog should appear. While this dialog is up you can continue to interact with the menus, tools, and windows of *WMS*. As you select different basins, the current parameters will be updated in the dialog.

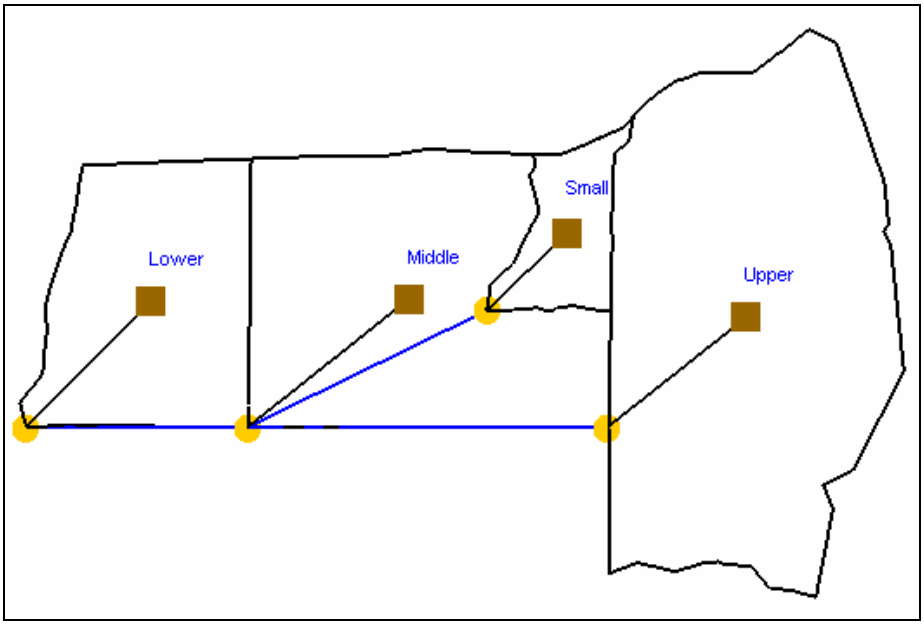


Figure 11-1 Basins.

11.2.1 Defining the Runoff Coefficients and Time of Concentration

The runoff coefficient, C, is used to account for losses between rainfall and runoff. The more developed a catchment is, the higher the C value it will have.

1. Enter a value of **0.20** for C

2. Enter a value of **22** for Time of Concentration

3. Select the basin labeled "Small" in Figure 11-1 by clicking on its basin icon or anywhere inside the basin (you may have to move the Rational dialog slightly in order to be able to select the icon, or you can close the rational dialog and then double-click the icon representing the basin).

4. Enter a value of **0.35** for C

5. Enter a value of **6** for Time of Concentration

6. Repeat this process for the other two basins, using the table below to fill in values for C and t_c

Basin Name	Runoff Coefficient C	Time of Concentration t_c
Upper	.2	22
Small	.35	6
Middle	.4	18
Lower	.4	11

7. When you are finished entering the parameters choose *Done* on the Rational Method dialog

A runoff coefficient coverage could be used to automatically map C values and basin data or a time computation coverage could be employed to determine Tc values, but they can also be computed/estimated separately and entered as demonstrated here.

11.2.2 Defining the Rainfall Intensity (i)

As part of the WMS interface to the Rational Method you can compute IDF curves using either HYDRO-35, NOAA, or user defined data. For this tutorial we will use HYDRO-35 data and a recurrence interval of 10 years.

1. Double-click the icon for the "*Upper*" catchment

2. Select the *IDF Curves…* button from the *Basin* section of the *Rational Method* dialog

3. Make sure the *HYDRO-35 Data (Eastern US)* radio group button is selected and select the *Define Data* button

4. Enter the following values to define IDF curves using HYDRO-35

Time (min)	Depth (in)
2 yr. 5 min.	.47
2 yr. 15 min.	.97
2 yr. 60 min	1.72
100 yr. 5 min.	.81
100 yr. 15 min.	1.75
100 yr. 60 min.	3.60

1. Select the *OK* button after correctly entering the rainfall values

The IDF curves for the 2, 5, 10, 25, 50, and 100 year recurrence intervals will be drawn, and values listed for selected times given in the windows on the right of the *IDF Computation* dialog.

1. From the text window in the upper right hand part of the dialog click on the line of data for the 10-yr recurrence interval as shown in Figure 11-2

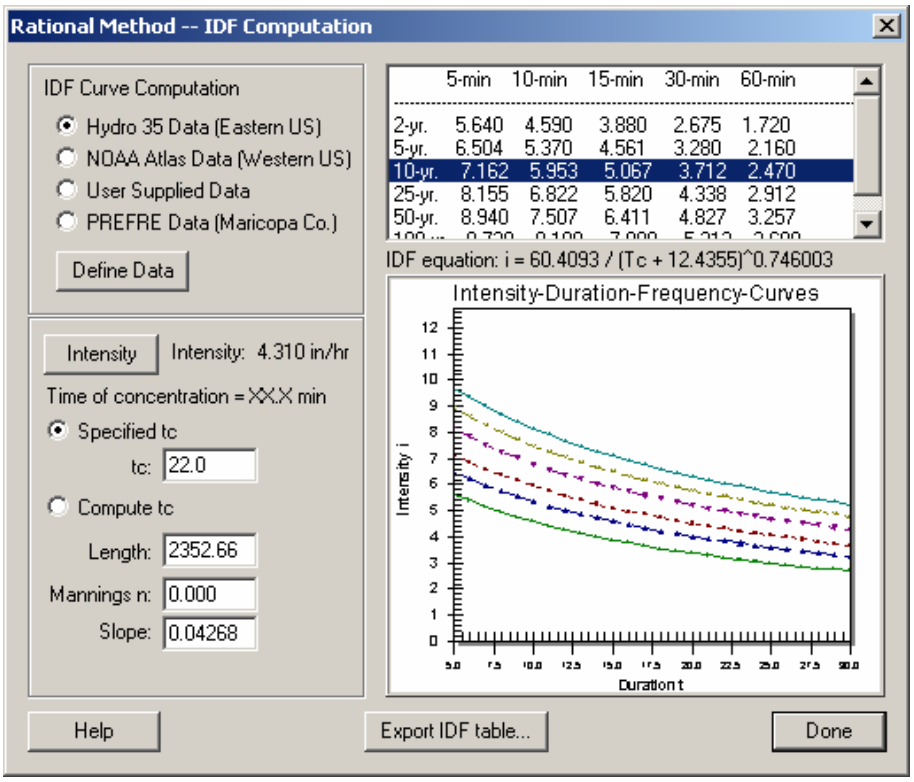


Figure 11-2 IDF Computation Dialog with the 10-yr Recurrence Interval Selected.

The rainfall intensity is determined from the selected interval by using the previously defined value for time of concentration.

1. Compute i by selecting the *Intensity* button

2. Select the *Done* button. The value of i computed using the *IDF-Computation* dialog will be placed in the edit field for this basin.

3. Note also that the input for this basin is complete and a value for runoff Q has been computed. A summary of the rational equation and results is in the upper right window of the dialog.

The HYDRO-35 data only needs to be entered once (unless different data is to be used for different basins), so the rainfall intensity for the remaining basins can be defined using the following steps.

1. Select the icon for "*Small*"

2. Select the *IDF Curves* button

3. Select the line of text for the *10-yr* recurrence interval

4. Select the *Intensity* button

5. Select *Done*

6. Repeat these steps for the Middle and Lower basins

7. When you are finished entering the parameters choose *Done* on the Rational Method dialog

11.2.3 Defining Hydrographs

As the data entry for each basin is completed, a peak flow (Q) is computed and listed in the text display window. The Rational Method equation does not produce a hydrograph. However, several different unit-dimensionless hydrographs can be used to distribute the peak flow through time to create a runoff hydrograph.

1. Double-click the basin labeled "*Upper*" in Figure 11-1

2. Select the *Define Hydrographs…* button

3. Select the *Rational method hydrograph* from the drop down list

4. Select *Done* to compute the hydrograph

5. Select *Done* for the Rational Method dialog

6. Double-click on the small hydrograph box to the upper right of the basin icon to open up a plot window with the hydrograph

You should see the hydrograph displayed in a plot window as shown in Figure 11-3.

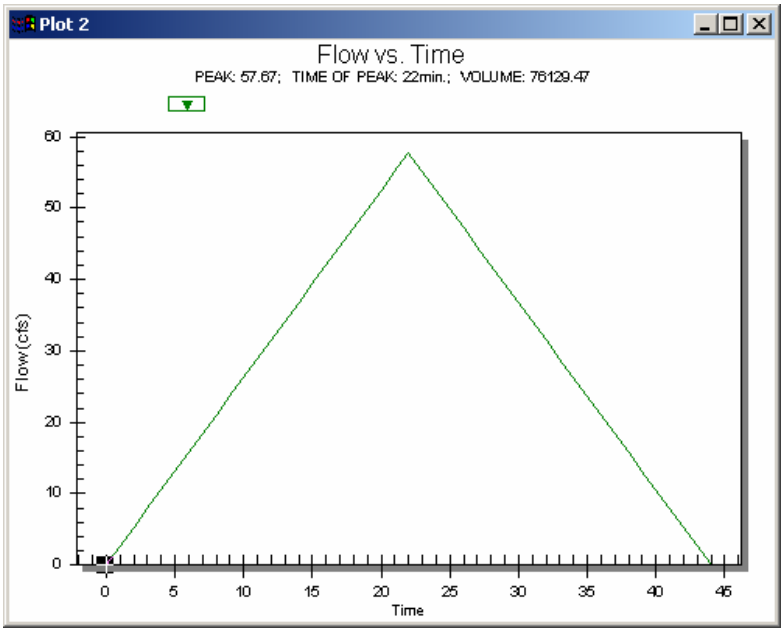


Figure 11-3 Rational Method Hydrograph for a Basin.

1. When you are finished viewing the hydrograph close the plot window by selecting the X in the upper right corner of the window

11.2.4 Defining Rainfall Intensities at the Outlet to Compute Runoff Hydrographs

WMS can determine composite rational method parameters at the outlet for computing hydrographs. The time of concentration at an outlet point is defined as the longest flow time from upstream basins (times of concentration) combined with any lag times from channels. The area is the cumulative upstream area, and the runoff coefficient is determined as an area weighted value from the upstream basins. With the time of concentration at the outlet defined you will need to determine the appropriate rainfall intensity. In order for WMS to compute peak flows and hydrographs at outlets you will need to define the travel time between outlets and the intensities for the time of concentration at each outlet.

1. Select the *Select Outlet* tool

2. Double-click the outlet icon of *Upper* as shown in Figure 11-1 (be sure to select the circular outlet icon and not the square basin icon).

You will note in the Outlet portion of the Rational Method dialog that information upstream from this outlet has been aggregated (in the case though there is just one basin upstream). The longest flow time is listed for the time of concentration, a cumulative area, and a weighted C value as shown in Figure 11-4.

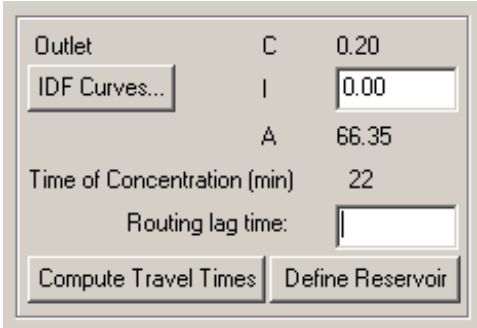


Figure 11-4 Outlet Information for a Rational Method Simulation..

1. Enter a value of **5** minutes for the *Routing lag time*
2. Select the *IDF Curves…* button
3. Select the line of text for the *10-yr*-recurrence interval
4. Select *Done*
5. Select the outlet icon of *Small*
6. Enter a value of **3** minutes for the *Routing lag time*
7. Select the *IDF Curves…* button
8. Select the line of text for the *10-yr*-recurrence interval
9. Select *Done*
10. Select the next downstream outlet (the outlet icon of *Middle*)

Note that for this outlet the upstream areas are summed, the C values weighted, and the longest travel path computed from the upstream basin Tc's and travel times between outlets determined.

1. Enter a value of **4** minutes for the Routing lag time
2. Select the *IDF Curves…* button
3. Select the line of text for the *10-yr*-recurrence interval
4. Select *Done*

The last or bottom-most outlet does not need to have a Routing lag time defined since the hydrograph accumulations will occur at this point, but you will still need to define the rainfall intensity.

1. Select the bottom-most outlet point

2. Select the *IDF Curves…* button
3. Select the line of text for the *10-yr*-recurrence interval
4. Select *Done*
5. Select the *Define Hydrographs…* button
6. Choose the Traditional method
7. Choose the *Rational Method hydrograph* option
8. Select *Done*
9. Select *Done* in the Rational Method dialog also
10. Double-click on the hydrograph icon for the most downstream outlet
11. Close the hydrograph plot window when you are done viewing by selecting the X in the upper right corner of the window

11.2.5 Combining Runoff from Multiple Basins

Besides the traditional method of computing peak flows and hydrographs for multiple sub-basins within a watershed, WMS will also allow you to lag hydrographs computed for basins and add at outlets in order to produce downstream peak flows and hydrographs.

1. Double-click on the downstream-most outlet point (outlet of the Lower basin)
2. Select the *Define Hydrographs…* button
3. Choose the *Route by summing* method
4. Choose the *Rational method hydrograph* option
5. Select *Done*
6. Select *Done*
7. Double-click the hydrograph icon for the bottom-most outlet

You can now see the difference between these two methods as both hydrographs are plotted in the window.

1. Close the hydrograph plot window when you are done viewing by selecting the X in the upper right corner of the window

11.3 Adding a Detention Basin

If you compute runoff using the route by summing method then you can route hydrographs through detention basin structures defined at any of the outlet locations.

1. Double-click the outlet that defines the Small catchment in Figure 11-1
2. Select the *Define Reservoir* button

You will now define a hypothetical detention basin for the Small catchment from approximate geometric parameters. WMS can compute a storage capacity curve for a rectangular basin. You could also enter a pre-computed storage capacity curve.

1. Click the *Define*... button
2. Select the *Define Storage*... button
3. Select the *Known Geometry* option
4. Enter **200** feet for *Length*
5. Enter **300** feet for *Width*
6. Enter a *Depth* of **30** feet
7. Enter a *Side slope* of **2**
8. Leave the *Base elevation* at **0.0** (It will be assumed on-grade at the outlet location)
9. Select *OK*

You will now define a standpipe and spillway (weir) for outlet structures and WMS will compute the elevation-discharge relationship automatically. In addition to standpipes and weirs you can define low-level outlets, or you can enter a pre-computed elevation-discharge relationship.

1. Select the *Define Discharges*... button
2. Select the *Add Standpipe* button
3. Set the *Pipe diameter* to **4** feet
4. Set the *Standpipe elevation* to **15** feet
5. Select the *Add Weir* button
6. Set the *Weir length* to **20** feet

7. Set the *Weir elevation* to **25** feet
8. Select *OK*
9. Select *OK*
10. Select *OK*

You have now defined a detention facility that has a standpipe and a spillway for control structures. The incoming hydrograph to this outlet point will be routed through the detention facility before being routed downstream and combined with the hydrographs of other basins.

1. Select the bottom-most outlet point
2. Select the *Define Hydrographs…* button
3. Choose the *Route by summing* method
4. Choose the *Universal hydrograph* method
5. Select *Done*
6. Select *Done*
7. Double-click on the hydrograph box for the bottom-most outlet

11.4 Conclusions

In this tutorial you have learned some of the options available for using the rational method in *WMS*. You will want to continue experimenting with the different options so that you can become familiar with all the capabilities in *WMS* for doing Rational Method simulations.

CHAPTER 12

National Flood Frequency Program (NFF) Interface

The National Flood Frequency program, developed by the USGS, provides a quick and easy way of estimating peak flows for ungaged watersheds. This data can be used in the design of culverts, flood-control structures, and flood-plain management. It utilizes regression equations that have been developed for each state. Most regression equations are functions of parameters such as area, slope, and runoff distance that are automatically computed by WMS when delineating a watershed.

In this tutorial you will compute peak flows for a watershed in Florida. An important part of runoff prediction in this region is the percent coverage of water bodies and so this is reflected in many of Florida's equations. There is not a direct calculation method available in WMS to determine the percentage of water-covered area in a watershed, but you will learn how you can use GIS overlay tools to compute percentages for any land use type within a watershed. This same method could be applied for equations in other states that require a percentage of water body coverage, or forested areas, or some other land use or soil formation that can be obtained from a shapefile or other GIS layer useable by WMS.

12.1 Opening the Drainage Basin

First we will open a WMS Project file (*.wpr) that contains a DEM that was previously downloaded from the Internet. A single watershed basin has been delineated from the DEM data and converted to feature objects.

Switch to the *Drainage* module

Select the ***File*** | ***Open…***

Open "*NFF_FL.wpr*"

12.2 Prepare the Basin for Use with NFF

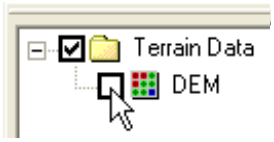
We will now use WMS to calculate the basin area, basin slope, and other parameters that can be used in conjunction with NFF.

1. Select ***DEM*** | ***Compute Basin Data***
2. Select the *Current Coordinates* button
3. Set the Horizontal and Vertical units to *Meters*
4. Select *OK*
5. Set the Basin Areas to *Square miles*
6. Set the Distances to *Feet*
7. Select *OK* to compute the parameters

In order to see the parameters that will be used with the NFF program, you can turn them on for display.

1. Select ***Display*** | ***Display Options…***
2. Select the *Drainage Data* tab
3. Check the display toggle for *Basin slopes* (*Areas* should already be on)
4. Select *OK*

Basin attributes are displayed at the centroid of the basin. In order to see the parameters more clearly, turn off the display of the DEM



5. If needed, expand the Terrain Data folder in the Data Tree
6. From the Data Tree, toggle off the check box for the DEM

Your screen should now look like Figure 12-1

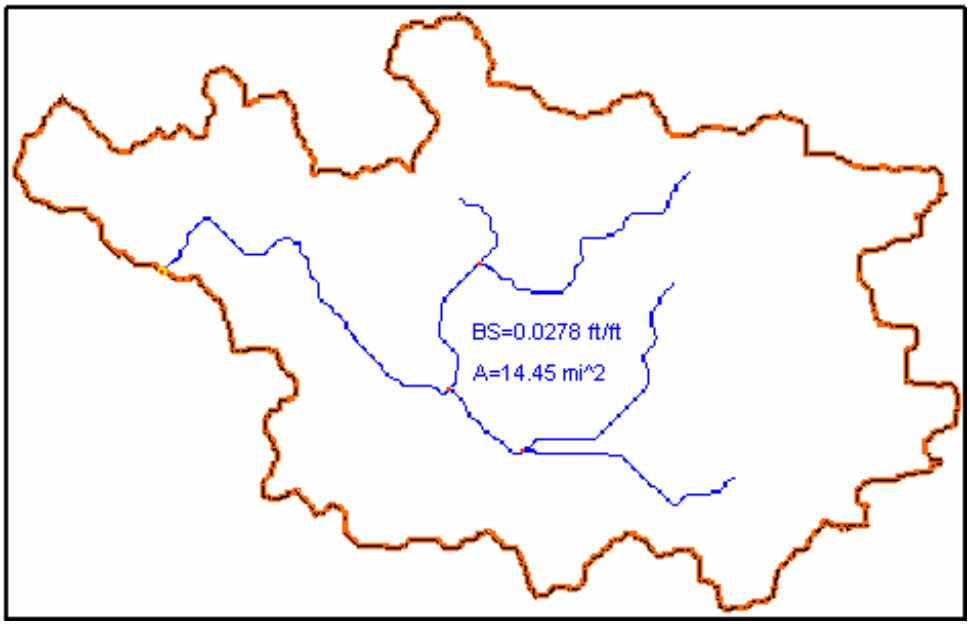


Figure 12-1: Drainage Basin with parameters computed

12.3 Calculating Percentage of Lake Cover

The regression equation for Region B of Florida includes a parameter (LK) to define the ratio of the area of lakes in the basin to the total basin area (as a percent). We will use the *Compute Coverage Overlay* calculator in WMS to calculate the percentage of lake cover in our drainage basin. The only other parameter in the regression equation for Region B of Florida is drainage area (DA), something that is automatically computed using the *Compute Basin Data* command.

12.3.1 Opening the Land Use Coverage

In order to compute the percentage of lake cover in our watershed, we will read in some land use data from a typical USGS land use file. Each polygon in the coverage is assigned a land use code that corresponds to a land use type. For this land use coverage, the codes for water bodies (lakes, reservoirs, wetlands) include 52, 53, 61, and 62. We will look for these codes to determine the value for LK.

1. Right-click on the *Coverages* folder in the Data Tree
2. Select *New Coverage*
3. Change the Coverage type to *Land Use*
4. Select *OK*

5. Switch to the *GIS* module
6. Select ***Data | Add Shapefile Data…***
7. Open "*valdosta.shp*"

This land use shapefile was obtained from www.webgis.com, but the EPA and other websites contain similar information. Alternatively, we could have digitized land use polygons from an image, as is discussed in Chapter 3 of the tutorials.

8. Choose the *Select Shapes* tool
9. Drag a selection box around the drainage basin polygon
10. Select ***Mapping | Shapes -> Feature Objects***
11. Select *Next*
12. The LUCODE with the land use ID is automatically mapped so you can continue by selecting *Next*

13. Select *Finish*
14. Hide *valdosta.shp* by toggling off its check box in the Data Tree (you may need to expand the GIS Layers folder to see it)

Only the portion of the shapefile that was selected will be used to create polygons in the Land Use coverage. The following figure displays the resulting land use polygons and their respective land use codes. This land use classification is consistent among all of the USGS land use data, were codes from 10-19 are urban, 20-29 agricultural, etc. A complete listing of code values can be found in the on line help file.

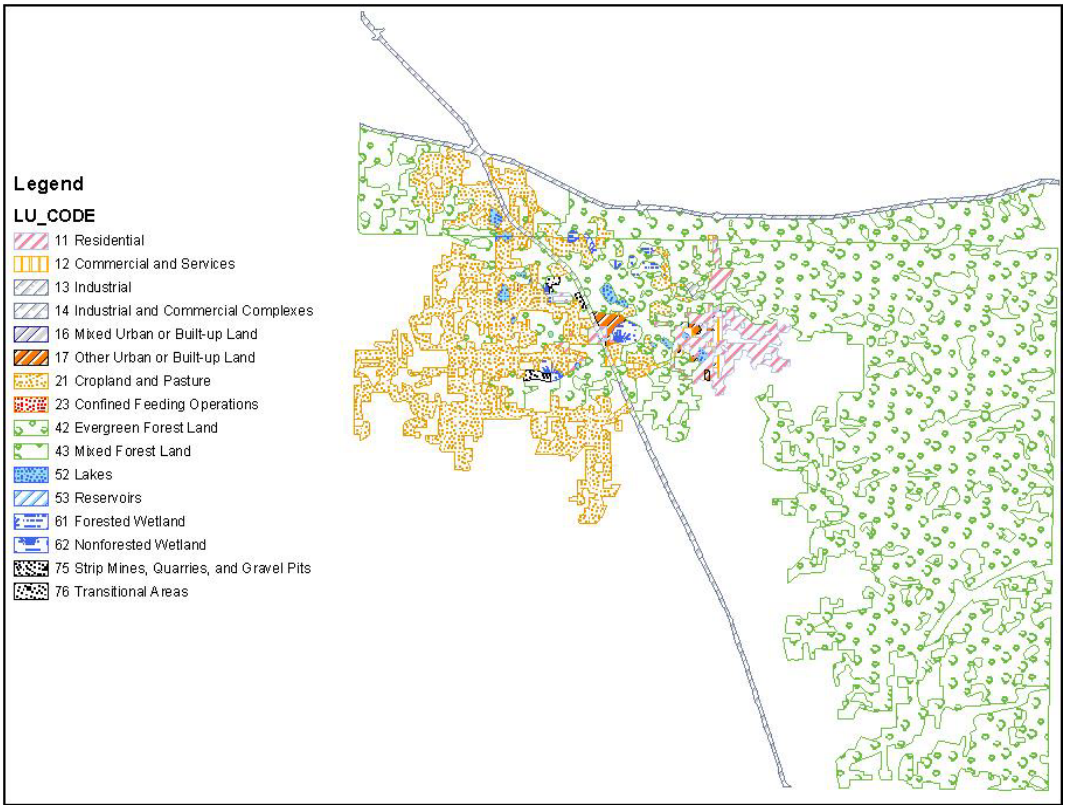


Figure 12-2: Land use codes used in Valdosta.shp.

12.3.2 Using the Compute Coverage Overlay Calculator

1. Switch to the Hydrologic Modeling module
2. Select ***Calculators*** | ***Compute Coverage Overlay…***
3. Make sure that *Drainage* is chosen as the Input Coverage
4. Make sure that *Land Use* is set as the Overlay Coverage
5. Select the *Compute* button

According to the USGS land use classification, code values in the 50's and 60's represent water bodies. To obtain the value for LK, we sum together the computed overlay percentages for Land Uses 52, 53, 61, and 62, as shown in Figure 12-3.

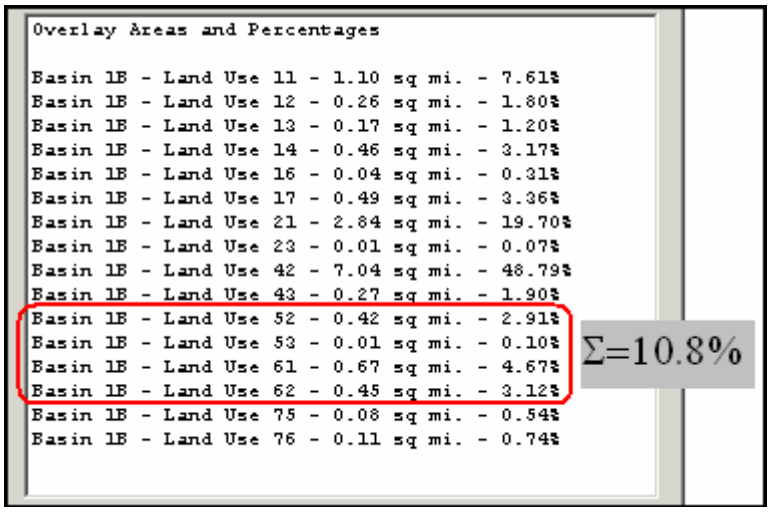

Overlay Areas and Percentages
Basin 1B - Land Use 11 - 1.10 sq mi. - 7.61%
Basin 1B - Land Use 12 - 0.26 sq mi. - 1.80%
Basin 1B - Land Use 13 - 0.17 sq mi. - 1.20%
Basin 1B - Land Use 14 - 0.46 sq mi. - 3.17%
Basin 1B - Land Use 16 - 0.04 sq mi. - 0.31%
Basin 1B - Land Use 17 - 0.49 sq mi. - 3.36%
Basin 1B - Land Use 21 - 2.84 sq mi. - 19.70%
Basin 1B - Land Use 23 - 0.01 sq mi. - 0.07%
Basin 1B - Land Use 42 - 7.04 sq mi. - 48.79%
Basin 1B - Land Use 43 - 0.27 sq mi. - 1.90%
Basin 1B - Land Use 52 - 0.42 sq mi. - 2.91%
Basin 1B - Land Use 53 - 0.01 sq mi. - 0.10%
Basin 1B - Land Use 61 - 0.67 sq mi. - 4.67%
Basin 1B - Land Use 62 - 0.45 sq mi. - 3.12%
Basin 1B - Land Use 75 - 0.08 sq mi. - 0.54%
Basin 1B - Land Use 76 - 0.11 sq mi. - 0.74%
Σ=10.8%

Figure 12-3: Summing the percentages of the codes representing water cover

The Coverage Overlay command can be used in a similar fashion to determine the percentage of forested areas (codes in the 40's), or any other classification type in a land use file, or a soil file.

1. Select *Done*

12.4 Running NFF

The geometric data computed from the DEM has automatically been stored with the NFF data. You can now run a simulation, using the derived data.

1. Make sure that the *Model* combo box is set to *NFF*
2. Select the *Select Basin* tool
3. Double-click on the basin icon for Basin 1B
4. Choose *Florida* from the list of states
5. Highlight *Region B* from the list of Regional regression equations
6. Select the *Select* → button to move Region B to the *Selected Equations* window
7. Enter **10.8** for the Lake Area variable (you may have to scroll the Variable Values window in order to see the Lake Area variable)
8. Select the *Compute Results* button

The peak Q values are displayed in the window at the bottom of the dialog.

12.4.1 Exporting the Flow Data

Once flow data is computed it may be exported to a text file in the format shown in the window, along with pertinent information used in computing the peak flow values.

1. Select the *Export* button
2. Locate a directory, and define a name for the file.
3. Select *Save*

The exported file can be viewed using any word processor, or inserted into a separate report document.

12.4.2 Generating a Hydrograph

The NFF program provides a way to determine an "average" hydrograph based on the computed peak flow and a basin lag time. A dimensionless hydrograph is used to define a basin hydrograph for the watershed based on the computed peak flow.

1. Scroll down in the Results window if necessary, and select the line of text corresponding to a Recurrence [years] of *50*
2. Select the *Compute Hydrograph* button.
3. Select the *Compute Lag Time – Basin Data* button.
4. Change the *Method* combo box to the *Custom Method* (the very last one in the list)
5. Select *OK*

The computed lag time in minutes is shown in the lag time edit field.

1. Select *OK*.
2. Select the *Done* button

A hydrograph icon will appear next to the basin icon for Basin 1B. You can examine the hydrograph in more detail:

1. Select the *Select Hydrograph* tool
2. Double-click on the hydrograph icon

The hydrograph is displayed in the Graphics window.

3. Close the hydrograph plot window by clicking on the X in the upper right corner of the window

12.4.3 Saving and Reading Simulations

You may wish to save this NFF simulation containing a state, selected regions, and defined variables. If you are running a demo version of WMS you should skip this step.

1. Select the ***NFF*** | ***Save Simulation…***
2. Save the simulation to your computer

The simulation may then be restored at a later date using the *Read Simulation* command in the *NFF* menu.

3. Select ***File*** | ***New***
4. Select *NO* when asked if you want to save your changes
5. Switch to the *Hydrologic Modeling* module
6. Select ***NFF*** | ***Read Simulation…***
7. Open the NFF file that you saved previously
8. Double-click on the icon for Basin 1B

Notice that although a simple tree network now represents your watershed, all of the needed variables computed previously (basin area, lake cover) is included in the NFF dialog.

9. Select *Done*

12.5 Utilizing an NFF Region Coverage

The NFF Region coverage type allows WMS to automatically determine which regression equations to use for an NFF simulation. Additionally, if a drainage basin overlaps multiple NFF regions, the NFF Region coverage automates the calculations for the percentage of the watershed in each region.

You will now use an NFF Region coverage to automatically assign the region for an NFF simulation. This coverage was digitized from an image that displays the NFF regions of Florida. This image was obtained from the NFF documentation and you could make a similar map by scanning (or capturing if

electronic) a map of the regions to create an image file, registering the image to a recognized coordinate system, digitizing the polygons (most states have less than 10 regions so it would take only a few minutes to digitize), and assigning the state and region. Details on how to scan images and create polygons by digitizing are given in the tutorials on images and feature objects. The USGS website for NFF has images available in the state by state documentation of the equations that can be saved directly and then registered in WMS. In this example we will read in an image that has already been registered and polygons that were digitized from the image.

1. Select ***File*** | ***New***
2. Switch to the *Map* module
3. Select ***File*** | ***Open…***
4. Open "*NFFmap_FL.jpg*"

If the image is opened into the registration dialog do the next three steps, otherwise skip them.

5. Select the *Import World File* button
6. Open "*NFFmap_FL.jgw*"
7. Select *OK*

The image is registered using the three cross hairs with latitude and longitude values and the coordinate system converted to UTM NAD 27 Zone 17 in order to overlay with the DEM and Land Use Data. For more details about how to use WMS to register an image, see the tutorial on Images.

12.5.1 Assigning Regions to Feature Polygons

We see from the image that Florida has three regions: A, B, and C. You will now open a WMS .map file that contains these three polygons in an NFF Region coverage. In the interest of time the polygons have already been digitized (see the tutorial on feature objects for more information about digitizing/building polygons), but the assignment of attributes (state/region) has been left for you to do.

1. Select ***File*** | ***Open…***
2. Open "*NFFmap_FL.map*"

This file was digitized directly from the image. See *Chapter 3: Basic Feature Objects* for information on how to digitize features from images.

3. Choose the *Select Feature Polygon* tool

4. Double-click on the polygon corresponding to the region labeled *C,* as shown in Figure 12-4

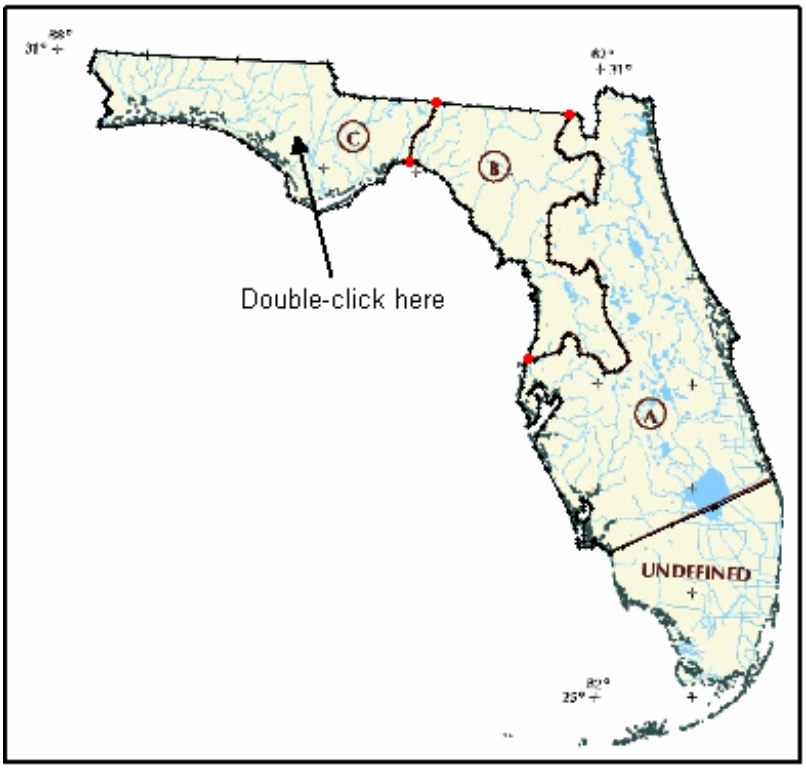


Figure 12-4: Image displaying NFF Regions for Florida

5. Choose *Florida* from the State list

6. Choose *Region C* from the NFF Region list

7. Select *OK*

8. Assign NFF Regions for the remaining two polygons in the same manner

12.5.2 Opening the Watershed

1. Select ***File*** | ***Open…***

2. Open "*NFF_FL.wpr*"

At this point, the study area appears as a small polygon. You will zoom in to better distinguish the area.

3. Choose the Zoom tool
4. Zoom in on the region indicated in Figure 12-5

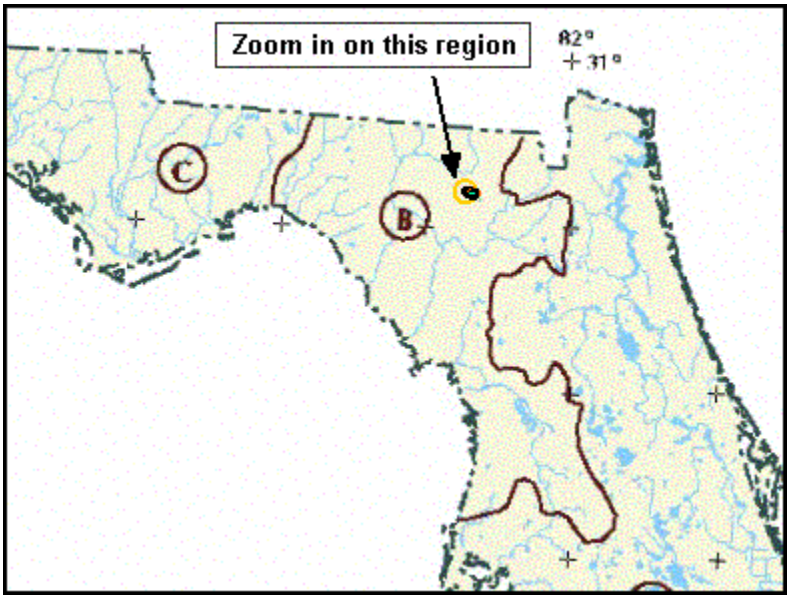


Figure 12-5: Zoom in on the watershed.

5. Select ***Display | Display Options…***
6. Choose the *DEM* tab
7. Toggle off the check box for displaying *DEM Contours*
8. Select *OK*
9. Select ***Feature Objects | Compute Basin Data…***
10. Select the *Current Coordinates* button
11. Make sure the Horizontal and Vertical units are set to *Meters*
12. Select *OK* twice

12.5.3 Running NFF

1. Switch to the *Hydrologic Modeling* module
2. Make sure Model combo box is set to *NFF*
3. Choose the *Select Basin* tool

4. Double-click on the icon for Basin 1B

5. Select *Yes*

Notice that the regression equation is automatically selected. Also, if our basin had overlapped with another NFF region, the areas and percentages of overlap for each region would also have been calculated.

6. Enter **10.8** for the Lake Area variable

7. Select the *Compute Results* button

8. Select *Done*

As you can see, the NFF Region coverage allows WMS to automatically load the appropriate regression equation(s) when we open the NFF dialog. Now, this might not save us a great deal of time if we are only running the simulation once for a single basin. However, if we plan to study many different basins on a regular basis, then creating an NFF Region coverage for our state would prove to be very efficient.

12.6 Conclusions

In this tutorial, we have discussed the following concepts in conjunction with setting up an NFF simulation:

1. How to calculate important parameters with the Compute Basin Data command

2. How to use the Compute Coverage Overlay calculator to compute areas of overlap between our drainage basin and any other coverage type

3. How to use an NFF Region coverage to automatically determine which equations should be used and to compute any areas of NFF region overlap

CHAPTER 13

HEC-RAS Analysis

HEC-RAS was developed by the U.S. Army Corps of Engineers Hydrologic Engineering Center. HEC-RAS performs a step backwater curve analysis for either steady state or transient conditions to determine water surface elevations and velocities.

13.1 Objectives

In this tutorial, you will learn how to use the WMS River tools to construct a HEC-RAS steady flow model. This includes the following:

- Building the conceptual model
- Mapping the conceptual data to a hydraulic model representation
- Running the simulation within HEC-RAS
- Viewing results in WMS

13.2 Preparing the Conceptual Model

The first step to creating a HEC-RAS model is to create a conceptual model which defines the river reaches (layout and attributes), the position of cross-sections on those reaches (orientation and station values), bank locations, and

material zones. The conceptual model will be used to create a network schematic inside the *River Hydraulic* module .

We will create the conceptual model from a USGS quad map as well as scattered bathymetric (elevation) data in the form of a TIN.

1. Select ***File*** | ***Open…***
2. Open "*wmsras.img*"
3. Select ***File*** | ***Open…***
4. Open "*wmsras.tin*"

The TIN clutters the screen, yet we want to know where it is so we don't create our conceptual model outside the domain of our bathymetric data. To better see the image, we will turn off the display of TIN triangles, vertices, and contours, and turn on the TIN boundary. To do this:

5. Select the *Display Options* button
6. Select the *TIN* tab
7. Uncheck the *Unlocked vertices* toggle box (it may already be unchecked)
8. Uncheck the *Triangles* toggle box
9. Uncheck the *TIN Contours* toggle box
10. Make sure the *Boundaries* box is checked
11. Select *OK*

13.2.1 Creating the Coverages

We need to create a centerline coverage for our reaches and a cross-section coverage for our cross-sections. These will form the core of our conceptual model.

1. Expand the *Map Data* folder in the Data Tree if necessary
2. Right-click on the *Coverages* folder within the Data Tree and choose *New Coverage*
3. Set its coverage type to *1D-Hyd Centerline*

4. Select *OK*

5. Create another new coverage and set its coverage type to *1D-Hyd Cross Section*

6. Select *OK*

7. Activate the *centerline* coverage by single-clicking on its name in the Data Tree

13.2.2 Creating the Centerline and Bank Arcs

Centerline arcs are used to define the locations and lengths of the study reaches and assign their attributes. We will have a centerline following the main channel as well as the tributary on the west. As the flows below the reservoir in the tributary on the East of the Leith Creek are small, we will disregard that reach in our simulation. To create the centerline arcs:

1. Select the *Create Feature Arc* tool

2. Following the pattern in Figure 13-1, create the centerline of the main channel from upstream to downstream (HEC-RAS views a river in this direction and it will ultimately help define what is the left bank and right bank characteristics) by clicking points on the centerline one at a time. Double-click the last point to indicate that it is the end of the centerline.

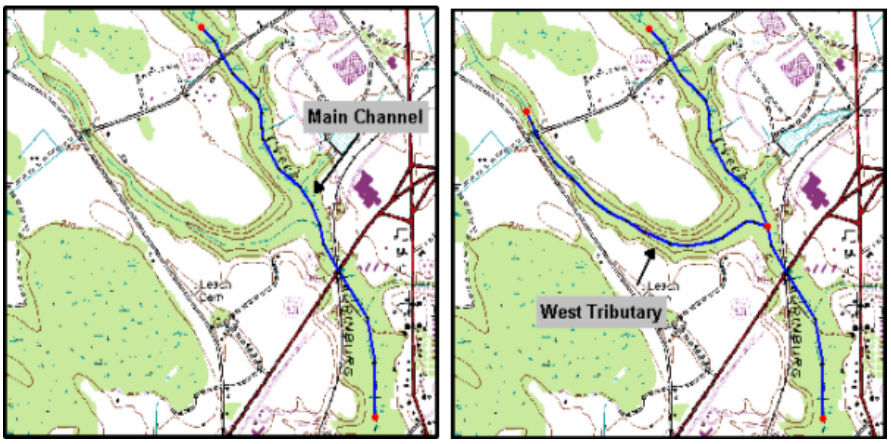


Figure 13-1: Creating the Centerline arcs

3. Create the arc for the west tributary, **upstream to downstream**, by clicking points on the centerline. Create the last point where the tributary meets the main channel by double-clicking on the main channel centerline. This splits the centerline of the main channel into two reaches.

This defines the centerline for the model in this simulation. It will consist of two reaches in the main channel (divided by the tributary), and one reach in the west tributary.

Bank arcs are used to define the locations of the banks and the over-bank distances. The next step is to create bank arcs along both sides of each centerline arc. To create the bank arcs:

1. Select the *Create Feature Arc* tool
2. Create new arcs where you estimate the bank locations to be, based upon contours/colors (roughly follow the green area around the centerline arcs) on the background image. Use Figure 13-2 as a guide.

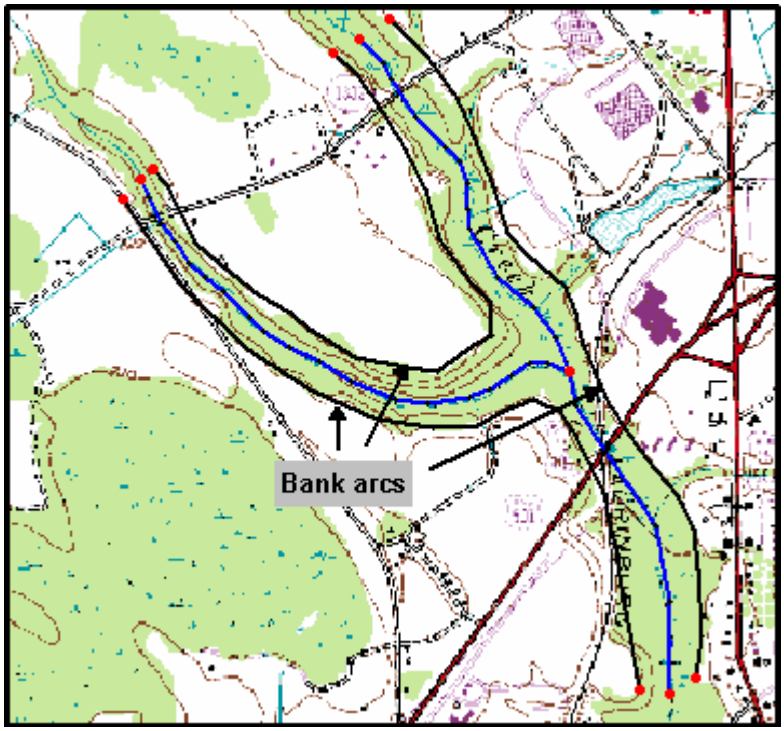


Figure 13-2: Placement of Bank arcs

1. Choose the *Select Feature Arc* tool
2. Select all of the bank arcs (hold the <Shift> key down while selecting in order to multi-select arcs)
3. Select ***Feature Objects | Attributes…***
4. Change the Arc Type to *Bank*
5. Select *OK*

The background image is no longer necessary. To turn it off:

1. Uncheck the toggle box next to the wmsras.img line in the Data Tree

13.2.3 Naming the Centerline Arcs

Reaches are stream sections where the flow rates and other hydraulic conditions are assumed to be constant. A river can be comprised of one or more reaches, but only one flow path. HEC-RAS has the ability to model multiple rivers (flow paths). To assign names to our rivers and reaches:

1. Double-click the uppermost reach in the main channel
2. Make sure the Arc type is set to *Centerline*
3. Select *OK*
4. Enter **Leith Creek** for the *River Name*
5. Enter **Upper Main** for the *Reach Name*
6. Select *OK*
7. Repeat steps 1 to 4 for each reach in the map as shown in Figure 13-3. (Note: For the Lower Main reach you can choose *Leith Creek* from the river name combo box instead of typing it in).

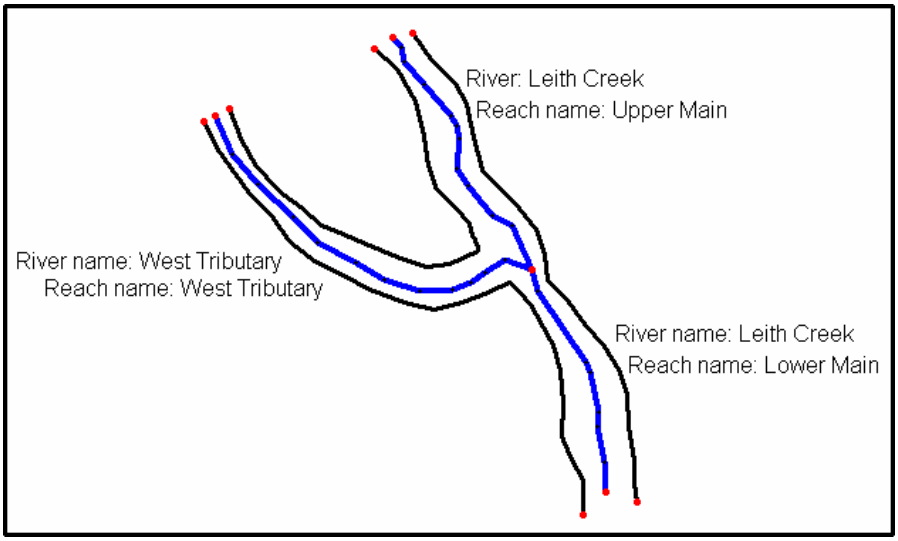


Figure 13-3: River and Reach Names

13.2.4 Creating Land Use / Materials Coverage

One of the properties HEC-RAS uses is roughness values. We will designate materials to different areas of our model. Later we will assign each material a roughness value. The material zones are stored in WMS as coverage type *Area Property*. To load the materials data:

1. Select ***File*** | ***Open…***
2. Open the file "*Materials.map*"
3. Select ***Edit*** | ***Materials…***
4. Click the *New* button 5 times in order to create 5 new material types
5. Rename the materials as shown below in Figure 13-4

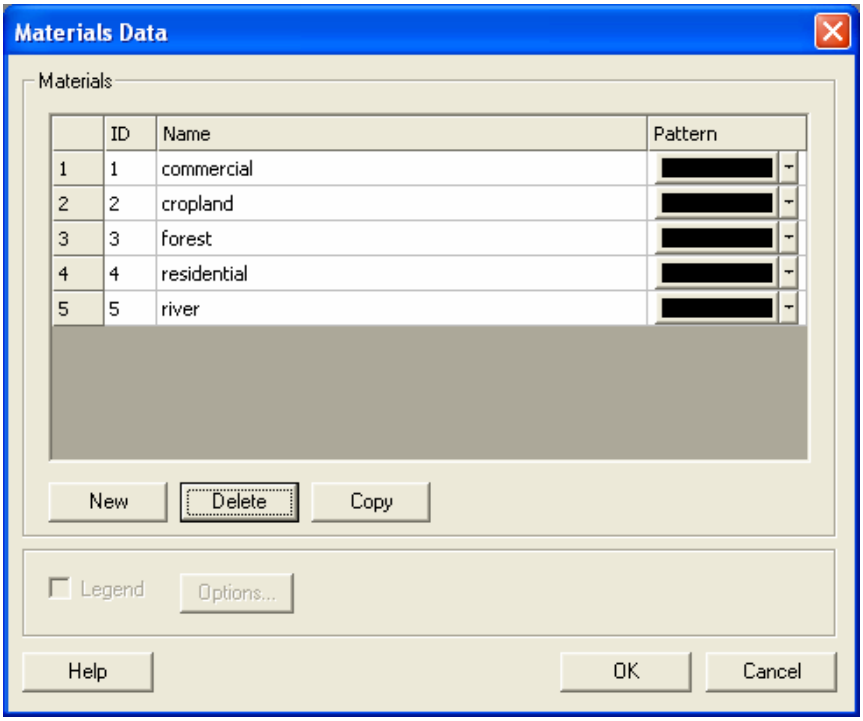


Figure 13-4: materials for use in the Area Property coverage

1. If you wish, you may set the color and pattern to better match the descriptions.
2. Select *OK* to close the *Materials Data* dialog
3. Make sure the newly created *Area Property* coverage "*materials*" is active in the Data Tree
4. Right click on the Materials layer and choose ***Properties…***

5. Change the *Coverage type* from *General* to *Area Property*.

6. Select *OK*

Now that you've defined all the materials, you will assign a material type to each polygon in the "materials" coverage.

7. Select the *Select Feature Polygon* tool

8. Select the polygon that defines the river area (see Figure 13-5)

9. Select ***Feature Objects | Attributes…***

10. Set the polygon type to *Material* and choose *river* from the list

11. Select *OK*

12. Using Figure 13-5 as a guide, define material types for the remaining polygons (remember you can also double-click on a polygon to bring up the attributes dialog)

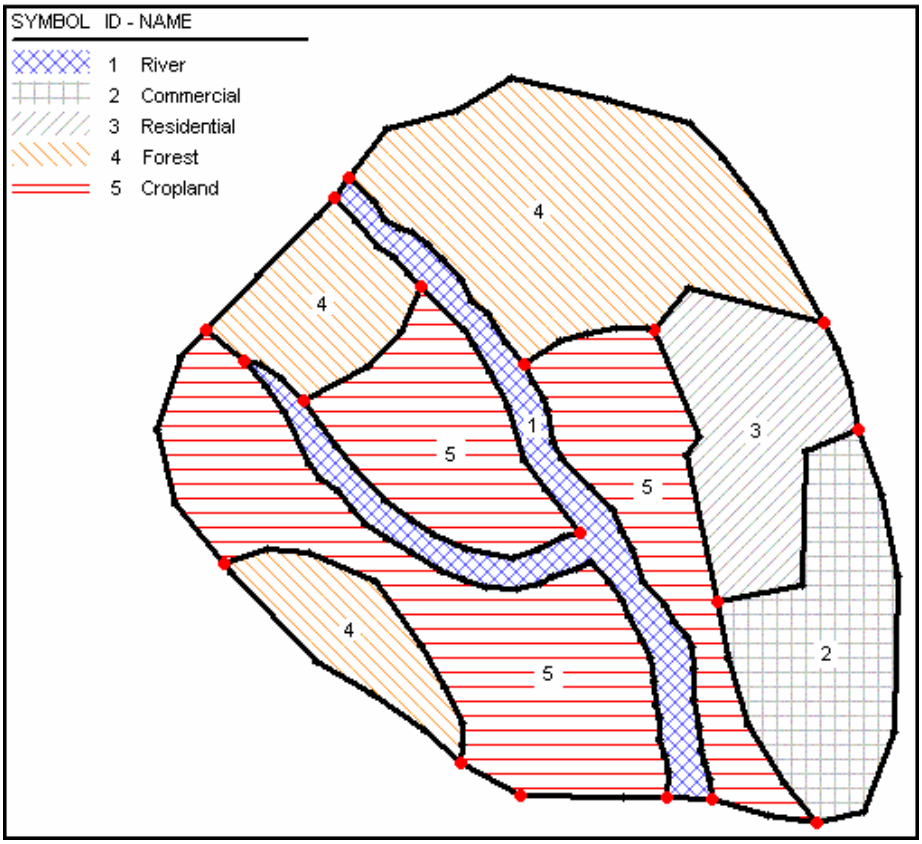


Figure 13-5: Materials used in HEC-RAS Simulation

13.2.5 Creating the Cross-Sections

HEC-RAS associates most of its model data with cross-sections and generates solutions or output at the cross-sections. Therefore, cross-sections are the most important part of the map. *HEC-RAS* requires at least two cross-sections on each reach. To create the cross-sections:

1. Set the current coverage to *1D Hyd Cross-Section* by single-clicking on it in the Data Tree
2. Select the *Create Feature Arc* tool .
3. Create at least two cross-sections on each reach by clicking a point on one side of the reach then double-clicking a point on the other side of the reach as shown in Figure 13-6.

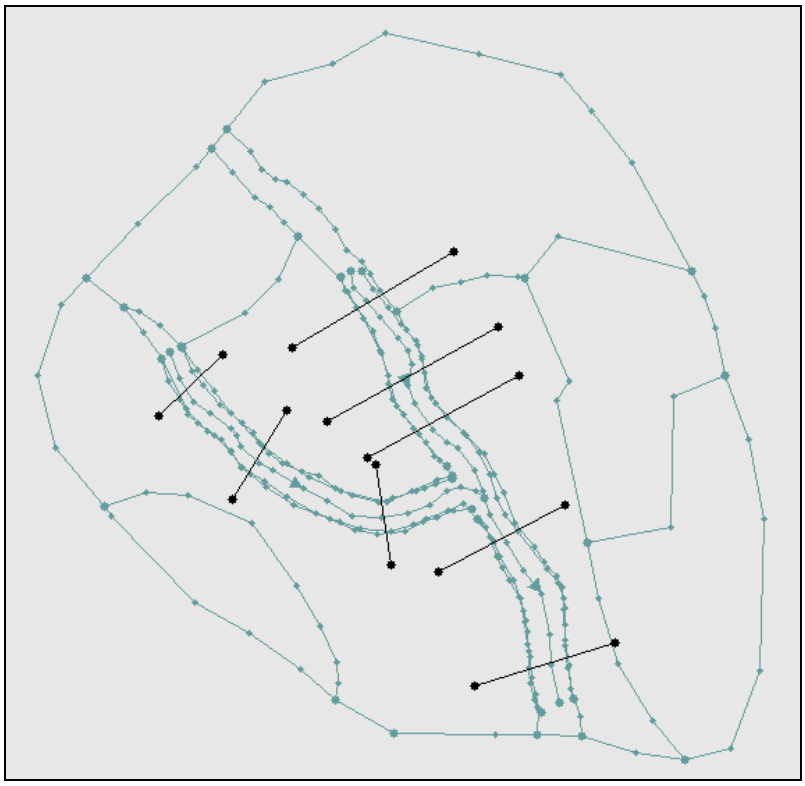


Figure 13-6: Cross-section coverage

13.2.6 Extracting Cross-Sections

In the cross-section coverage, all arcs are cross-section arcs. Their position and orientation define the location of the cross-sections in the system, but as of yet, they do not have any data assigned. We want to assign elevation data, materials, and point property locations to the cross-sections. This information

will be extracted from the TIN, the area property coverage, and the centerline coverage. To extract this data:

1. Select ***River Tools | Extract Cross Section…***

2. WMS will extract an elevation point at every triangle edge along the cross-section arc. The default extraction settings are to use the Centerline coverage to generate point properties and the Area Property coverage to define material zones.

3. Select *OK*

4. When WMS prompts you for filename and location for saving the cross-section database, enter the name **xsecs** and select *Save*

Each cross-section arc now stores a link to a cross-section database record which contains xyz data, materials properties, bank locations, and thalweg locations. To view and edit the information at a cross-section:

5. Choose on the *Select Feature Arc* tool

6. Double click on any cross-section. This brings up the *River Cross Section Attributes* dialog

7. Make sure that the reach name is assigned correctly

8. Click on the *Assign Xsec* button. This brings up the *Assign Cross Section Profile* **dialog**, which is used to view the current cross section shape and select a different cross-section from a cross-section database if desired

9. Click on the *Edit* button. This brings up the *Cross-Section Attributes* dialog. This dialog can be used to view and/or edit the cross-section

10. Click on the *Line Props* **tab** to view the materials that are assigned to the cross-section

11. Click on the *Point Props* **tab** to view the locations of the left bank, right bank, and thalweg

12. Select *Cancel* until all the dialogs are closed

For this tutorial we will use the cross sections as extracted from the TIN, but often for a project you will need to edit a cross section, or merge surveyed cross sections and other editing procedures. This is the topic of a separate tutorial and therefore is not covered in detail here.

13.3 Creating the Network Schematic

WMS interacts with HEC-RAS using a HEC-GeoRAS geometry file. This file contains the cross-sectional data used by HEC-RAS in addition to three dimensional georeferencing data. To create this geometry file the conceptual model must be converted to a network schematic diagram in the ***River Module***. To convert the conceptual model to a network schematic:

1. Make sure you are still in the *Map* module
2. Set the current coverage to *centerline* by clicking on it in the Data Tree
3. Select ***River Tools | Map -> 1D Schematic***
4. Your screen should appear similar to Figure 13-7

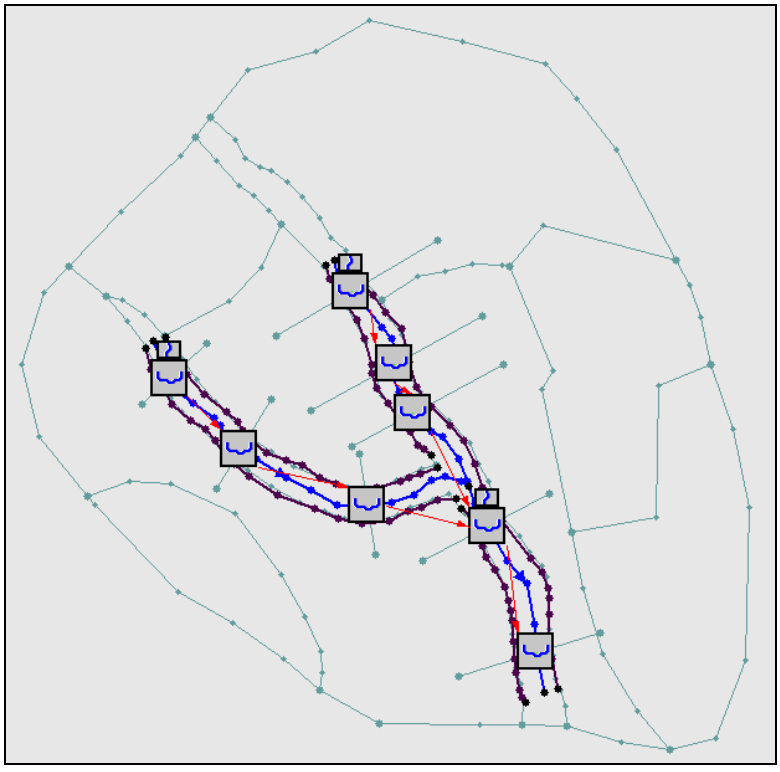


Figure 13-7: Schematic diagram

Now WMS includes two separate representations of the data. The first you created as a conceptual model, which is stored as a series of coverages. The second is a numeric model stored as a schematic of cross sections organized into reaches. Modifications to the network schematic that can be used by HEC-RAS can be made directly in the ***River Module***, or indirectly by editing the conceptual model in the ***Map Module*** and mapping to a new network schematic.

HEC-RAS needs Manning's roughness values for the materials found in the cross-section database. The roughness values are stored as part of the 1D model in the River Hydraulics Module. To specify the roughness values for the each of the materials:

1. Switch to the *River* Module .

2. Select ***HEC-RAS | Material Properties***

3. Enter the roughness values for each material as shown in Figure 13-8

Figure 13-8: HEC-RAS Material Properties Dialog

1. Select *OK*

Now we need to tell HEC-RAS which set of line properties in the database should be used as material types. To do this:

2. Select ***HEC-RAS | Model Control***

3. Select the line property name that stores the roughness values for the cross-section database. In this case, the line property is named Materials which came from our Area Property coverage *Materials*

4. Select *OK*

13.4 Creating the Geometry Import File

Now that the model has been setup, we need to create the geometry file. To create this file:

1. Select ***HEC-RAS | Export GIS File...***
2. Name the file **hecras.geo**
3. Select *Save*

After saving out the GIS file, WMS automatically opens the HEC-RAS application, loads the GIS file, and creates a new HEC-RAS project named "*hecras.prj*."

13.5 Using HEC-RAS

Within HEC-RAS, we will setup and run the simulation and then export the results for post-processing in WMS. The setup will include entering junction lengths, specifying flow values, and assigning river boundary conditions.

1. Select ***Edit | Geometric Data…***

Your screen should appear like Figure 13-9 below:

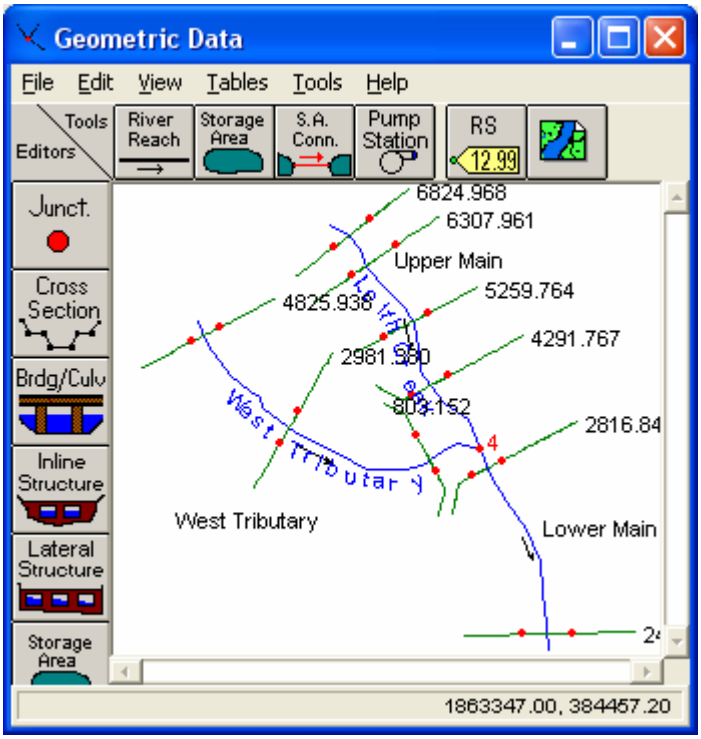


Figure 13-9: The Geometric Editor in HEC-RAS.

In the case that an excessive number of points are extracted from the cross-section arcs, HEC-RAS has a tool that will filter points that are too close together to run an analysis. Since there are many data editing tools available within HEC-RAS in addition to the filtering tool, we simply make you aware of such tools and suggest that you browse the HEC-RAS user's manual for more information.

First, we will enter junction lengths in the Geometric Data Editor.

1. Left-click on the node (junction) that joins the West Tributary and Upper Main reaches.

2. Select *Edit Junction…* from the pop-up menu

The Junction Length value should be entered as the distance stretching across the junction between adjacent cross-sections. We will measure these lengths with the Measure Tool in WMS.

3. Activate the WMS window

4. Switch to the *Map* module

5. Choose the Measure Tool

6. As shown in the example in [fig] below, trace out the distance along the centerline arc between the most downstream cross section arc on the West Tributary reach and the most upstream cross section arc on the Lower Main reach

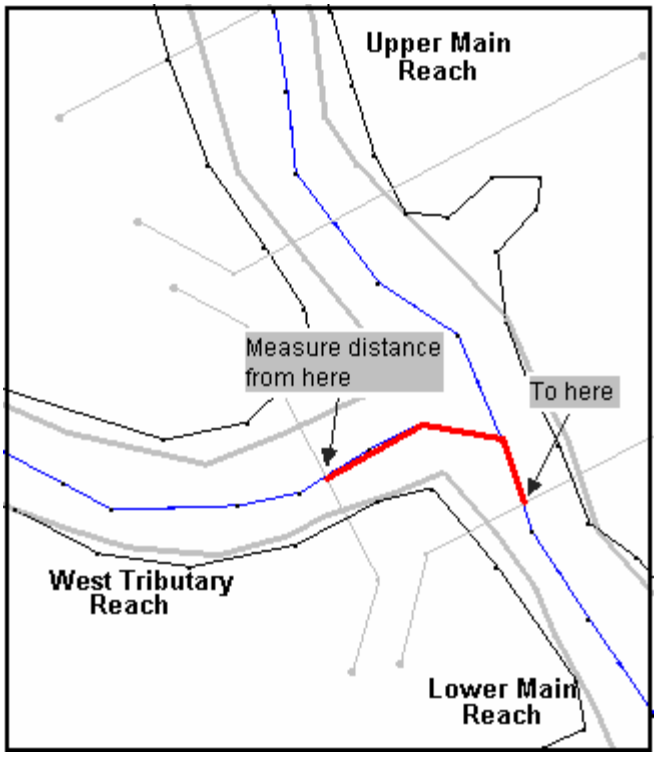


Figure 13-10: Measuring Junction Lengths.

7. Revert back to the HEC-RAS Junction Data dialog

8. Enter the measured length in the Junction Length field corresponding to the "To: West Tributary – West Tributary" line

9. Repeat this step to measure and enter the Junction Length between the most downstream cross section arc on the Upper Main reach and the most upstream cross section arc on the Lower Main reach

10. Once you have finished entering both lengths, select *OK*

11. Select ***File | Exit Geometry Data Editor***

The next step is to define the flow and boundary conditions for our reaches. To define this information:

1. Select ***Edit | Steady Flow Data***

2. For Profile 1 (PF 1), enter **4000** for Upper Main; enter **5000** for Lower Main; enter **1000** for West Tributary

3. Click on the *Reach Boundary Conditions* button

For our analysis we are going to have HEC-RAS compute normal depths at the boundaries of our model. To do this:

1. For each of the blank boxes in the spreadsheet, select the box and click on the **Normal Depth** button. Enter the following values for the slopes of each reach: **0.003** for the upper reach, **0.001** for the lower reach, and **0.005** for the tributary.

2. Select *OK*

3. Click the *Apply Data* button

4. Select ***File | Exit Flow Data Editor***

We are now ready to run the steady flow analysis. We first need to set an option to set flow distribution locations so that velocity profiles will be computed. To set this option and perform the analysis:

1. Select ***Run | Steady Flow Analysis*** from the menu

2. Select ***Options | Flow Distribution Locations***

3. Change the *Global subsections* to *3* in each of the three fields (LOB, Channel, and ROB)

4. Select *OK*

5. Click the *Compute* button. This runs the 1D analysis

6. Close the *Steady Flow Analysis* Dialog

7. Exit out of the *HEC-RAS* program

13.6 Post-Processing

Now that HEC-RAS has computed water surface elevations, we can read the solution into WMS. The water surface elevations are read in as 2D Scatter Points and can be used to perform a floodplain delineation. To read the solution:

1. In WMS, make sure you are in the *River* module

2. Select ***HEC-RAS | Read Solution…***

3. Open "*hecras.prj*"

4. In the Data Tree, expand the 2D Scatter Data folder, if necessary

Notice that a new scatter point file has been read into WMS. These points contain the water surface elevations computed by HEC-RAS. Since we used 7 cross-sections in our model, we only have 7 scatter points with which to

perform the floodplain delineation. In order to achieve a more accurate floodplain delineation, however, WMS has a tool that interpolates scatter points along centerline and cross-section arcs.

1. Switch to the *Map* module
2. Select the *1D-Hyd Centerline* coverage from the Data Tree
3. Select ***River Tools | Interpolate Water Surface Elevations…***
4. Note that *hecras* is selected as the Scatter dataset. In this case, it is the only scatter dataset available. But if you had multiple datasets, you could specify which one to use for the interpolation.
5. Choose *At a specified spacing* for the *Create a data point* option
6. Enter **60** for the spacing
7. Select *OK*
8. Select the *1D-Hyd Cross Section* coverage from the Data Tree
9. Select ***River Tools | Interpolate Water Surface Elevations…***
10. Select *OK*
11. Switch to the *Terrain Data* module
12. Select ***Flood | Delineate…***
13. Toggle on the *Search radius* check box and enter **1000** for the radius
14. Toggle on the *Quadrants* check box
15. Enter **4** for the *number of stages in a quadrant*
16. Select *OK*
17. WMS now computes two new datasets corresponding to floodplain depths and water surface elevations (river bed elevation + flood depth)
18. Expand the folder named New tin in the Data Tree
19. Expand the folder named *W.S. (FLOOD)* in the Data Tree
20. Select the data set named *W.S. Elev-PF 1 (FLOOD)*
21. Select ***Display | Display Options…***
22. Choose the *TIN* tab

23. Toggle on *TIN Contours* and choose the *Contours* button

24. Choose the *Color fill between contours* option

25. Select *OK* twice to exit both dialogs

These color-filled contours represent the varying water surface elevations in the computed floodplain. For more information and practice on flood plain delineation, see the chapter entitled Floodplain Delineation. A second HEC-RAS tutorial where surveyed cross sections are used can be found in chapter 14.

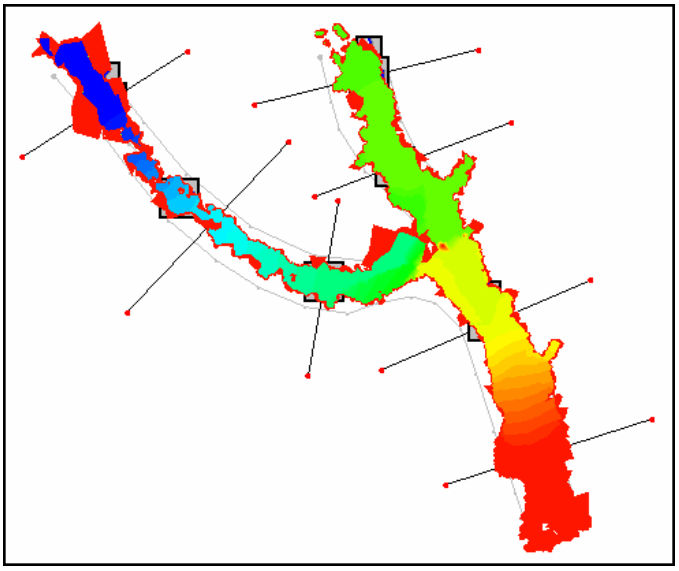


Figure 13-11: Resulting TIN from the Floodplain Delineation

CHAPTER 14

HEC-RAS – Managing Cross Sections

HEC-RAS is a 1-D river model that relies on cross section data along reaches to compute results. Cross sections can be extracted from a TIN in WMS, but the TIN does not always define the channel with enough resolution to get an accurate cross section through the channel. Tools in WMS make it possible to manage cross sections by editing their shape, defining properties, and merging multiple cross sections together. Surveyed channel cross section data can be merged with cross sections extracted from a TIN in order to develop cross sections that accurately depict both the channel and surrounding terrain.

14.1 Objectives

In this tutorial you will learn how to merge cross sections and use the data for input into a HEC-RAS model by:

1. Creating a conceptual river model
2. Converting a DEM to a TIN for background elevation data
3. Extracting cross sections
4. Merging cross sections
5. Running HEC-RAS
6. Delineating the floodplain using HEC-RAS results

14.2 Creating a Conceptual River Model

A conceptual river model can be created by using GIS data to define centerline and bank arcs, material properties, and cross section location and geometry. Background images are also useful in creating and viewing this GIS data.

14.2.1 Open Images

1. Switch to the Map module
2. Select ***File*** | ***Open…***
3. Open "*jordannarrows.jpg"*and *"midvale.jpg"*

14.2.2 Define Centerline and Bank Arcs

1. Select ***File*** | ***Open…***
2. Open "*river.map*"
3. Zoom in close enough around a section of the feature arcs so that you can see three distinct arcs as shown in Figure 14-1

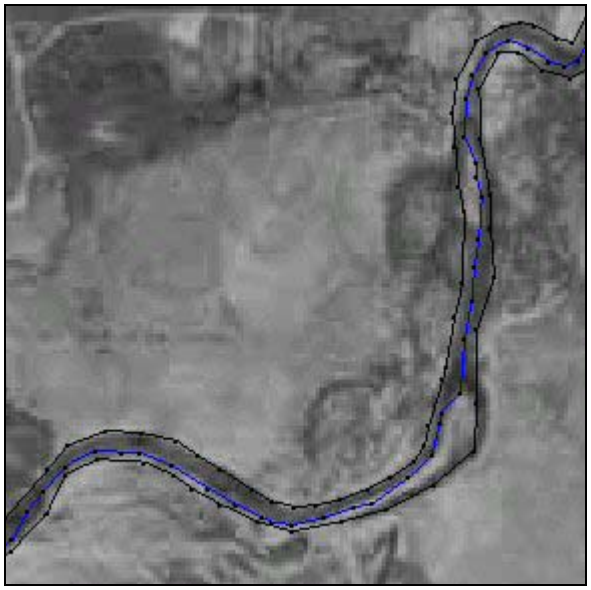


Figure 14-1: Centerline and bank arcs.

The middle arc is the centerline arc that defines the thalweg of the river reach and the outer arcs define the right and left banks.

4. Use the *Select Feature Arc* tool to select the middle arc
5. Select ***Feature Objects*** | ***Attributes…*** and choose *Centerline*
6. Select *OK*
7. Enter **Jordan River** for the *River Name*
8. Enter **Riverton** in the *Reach Name*
9. Select *OK*
10. Use the *Select Feature Arc* tool to select both outer arcs
11. Select ***Feature Objects*** | ***Attributes…*** and choose Bank
12. Select *OK*

14.2.3 Create Materials

Materials are defined on an *Area Property* coverage by digitizing polygons representing different materials using a background image, such as an aerial photograph, or by using land use data from a shapefile or another source. Each different material that is defined will be used later to assign roughness values to the cross sections. In this tutorial we will read in a *.map file containing material data that has already been digitized.

1. Select ***File*** | ***Open…***
2. Open "*materials.map*"
3. Zoom in around the materials polygons on the *Area Property* coverage
4. Select ***Display*** | ***Display Options…***
5. Toggle *Color Fill Polygons* on
6. Select *OK*
7. Select ***Edit*** | ***Materials…*** to view the material types associated with each color
8. Select *Cancel*

14.3 Converting a DEM to a TIN

Cross section geometry can be extracted from a TIN in WMS. We will create a TIN by opening a DEM and converting it to a TIN. It is also possible to filter redundant DEM points out of the TIN that we create.

14.3.1 Open DEM

1. Switch to the Terrain Data module
2. Select ***File*** | ***Open…***
3. Open *"91606647.hdr"*
4. Select *OK*
5. Select *Yes* to convert coordinates
6. Toggle *Edit project coordinate system* on
7. Set the Convert From Horizontal System to *Geographic NAD 83 (US)*
8. Set the Convert To Horizontal System to *UTM NAD 27 (US)*
9. Select *OK*

14.3.2 Convert to a TIN

1. Select ***DEM*** | ***Conversion*** | ***DEM->TIN*** | ***Filtered…***
2. Select *OK*
3. Select ***Display*** | ***Display Options…***
4. Toggle *Triangles* off and *Boundaries* on
5. Select *OK*
6. Expand the *Terrain Data* folder in the Data Tree

7. Hide the DEM by toggling its checkbox off

14.4 Extracting Cross Sections

It is very easy to extract cross section geometry from a TIN in WMS. This is done by creating arcs that represent the plan view of the cross sections on a *1-D Hyd Cross Section* coverage.

1. Switch to the Map module
2. Select ***File*** | ***Open…***
3. Open *"xsections.map"*
4. Zoom in around the cross section arcs labeled 0 - 4

5. Hide the materials polygons by toggling *Area* Property coverage in the Data Tree off
6. Hide the background images by toggling them off
7. Select ***Display*** | ***Contour Options…***
8. Select the *Specified interval* option and enter a value of **10.0**
9. Select *OK*

You can see that the cross section arcs are approximately as wide as the floodplain might be. They end where there is a sharp break in slope and the terrain gets relatively steep. Wider cross section arcs are generally not necessary for a HEC-RAS analysis.

10. Make sure the *1D-Hyd Cross Section* coverage is active in the Data Tree

11. In the Model drop-down list at the top of the screen select *River Tools*
12. Select ***River Tools*** | ***Extract Cross Section***…
13. Select *OK*
14. Enter **xsec** for the name of the new cross section database where all of the cross sections will be stored

14.5 Merging cross sections

Any two cross sections can easily be merged in WMS. We will create a new cross section database that stores surveyed channel cross section data and merge that with the cross sections that we just extracted from the TIN.

Merging cross sections will create more accurate cross section geometry data. Cross sections are merged by aligning both cross sections using reference points such as the thalweg or bank locations and then inserting points from one cross section into the other.

14.5.1 Open channel cross section data

1. Select ***File*** | ***Edit File…***
2. Open "*channel.txt*"
3. Select *OK* to open with Notepad or choose any other text editor or spreadsheet

14.5.2 Create a new cross section database

1. Select ***River Tools*** | ***Manage Cross Sections…***
2. Choose the *New Cross Section Database…* button
3. Enter **channelxsec** for the *File name*
4. Select *Save*

14.5.3 Define channel cross sections in the database

The current cross section database should be "*channelxsec.idx*" (the one that you just created).

1. Click on the *Edit Cross Section Database…* button
2. Click the *New* button to add a cross section to the database
3. Select the *Edit* button to edit the cross section
4. Click the *Add* button and enter **7** to add 7 points to the cross section
5. Select *OK*
6. Toggle *XY* off
7. Copy and paste the cross section data for Channel Section 1 from "*channel.txt*" (opened in Notepad) into the Cross-Section Attributes Geom Edit tab as shown in Figure 14-2. Make sure to paste the data into the first row.

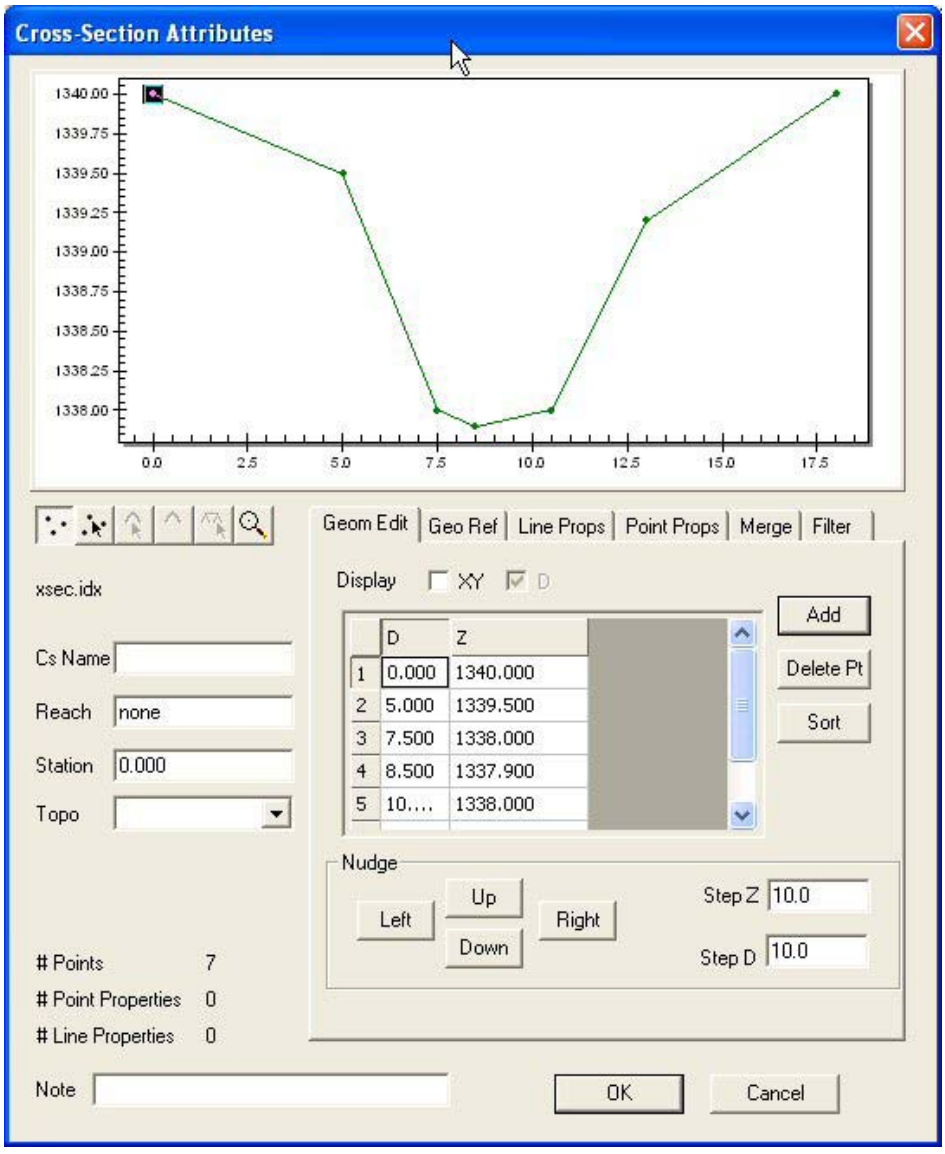


Figure 14-2: Adding cross section geometry data to the database.

8. Choose the *Point Props* tab

9. Click the *Auto Mark* button to automatically define thalweg and right/left bank points. These reference points will be used to align cross sections for merging.

10. Select *OK*

11. Repeat steps 2-10 to add the 4 remaining channel cross sections in "*channel.txt*" to the cross section database

12. Click the *Save* button

13. Select *OK*

14. Select *OK*

14.5.4 Align channel cross sections with extracted cross sections

1. Use the *Select Feature Arc* tool to select the cross section arc at the top of the screen labeled 0

2. Select ***Feature Objects*** | ***Attributes…***

3. Select *Assign Xsec*

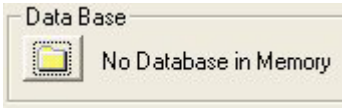
This will allow you to view all of the extracted cross sections and assign the highlighted geometry to this cross section arc.

4. Click the *Edit* button

5. Select the *Line Props* tab to view the material properties (roughness values) that will be applied to each cross section. These can be edited if necessary.

6. Select the *Merge* tab

7. Select the *Load Insert Cs* button



8. Click on the Data Base *Browse* button

9. Open "*channelxsec.idx*"

10. Make sure that the csid number that is highlighted for the channel cross section matches the csid number of the extracted cross section that you are editing (these numbers also match the number labels displayed next to each cross section arc on the screen)

11. Select *OK*

You will see a profile of the surveyed cross section geometry appear in the upper left corner of the profile view of the extracted cross section geometry as shown in Figure 14-3.

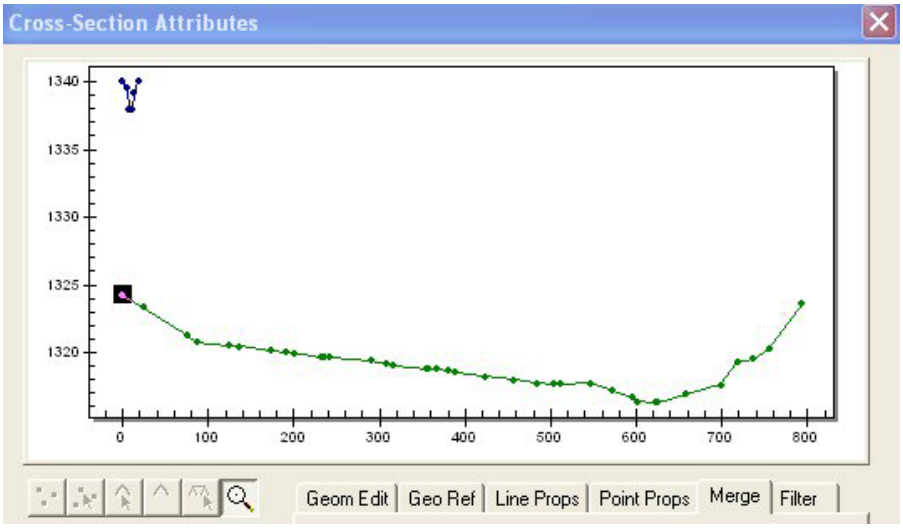


Figure 14-3: Merging cross sections.

Merge cross sections using the Alignment tools shown in Figure 14-4. You can align cross sections using reference points (point properties) that are defined on both cross sections such as left end, left bank, thalweg, right bank, and right end. Enter a value for the *Offset* the and cross section will be offset that distance from the alignment point. Specify a distance for *Step Z* and use the *Up* and *Down* buttons to move the inserted cross section vertically. Do the same for *Step D* using the *Left* and *Right* buttons to move the cross section horizontally. The horizontal and vertical scales on the plots are useful for determining the distances to enter. (Remember that the scales are not equal and so the vertical distances are magnified)

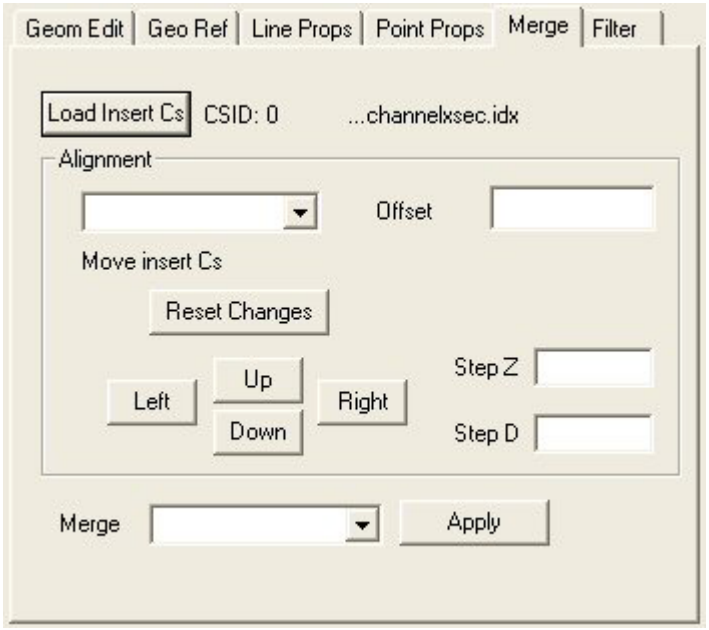
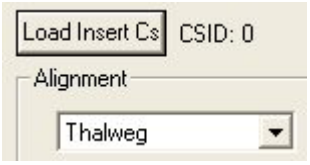


Figure 14-4: Alignment tools.



12. Choose *Thalweg* for Alignment to align the thalweg (specified as a point property) of the channel cross section with the thalweg of the extracted cross section

13. Enter a value for *Step Z* (try **5**) and use the *Down* button to move the channel cross section vertically

14. Reduce the *Step Z* value to **1** and use the *Up* button to position the cross section

15. Keep reducing the *Step Z* value and using the *Up* and *Down* buttons until the cross sections are aligned correctly

16. Enter a value for *Step D* and use the *Left* and *Right* buttons if you need to move the channel cross section horizontally

The Zoom tool is useful for viewing the alignment of the cross sections close-up once you have the channel cross section located in the general area where it will be inserted.

The aligned should look similar to Figure 14-5:

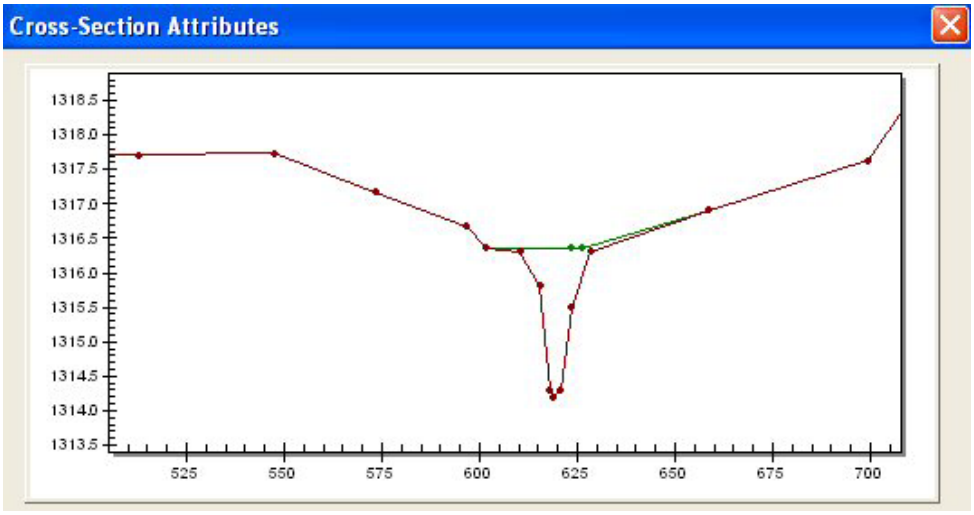


Figure 14-5: Aligned cross sections.

17. For Merge select *Insert All*

18. Click *Apply* to insert the channel cross section and permanently change the extracted cross section data

19. Select *OK* and notice the updated extracted cross section geometry

20. Select *OK*

21. Select *OK*

22. Repeat this process (steps 1-21) for all of the cross section arcs

14.6 Running HEC-RAS

A schematic will be created using the GIS data defined in WMS and exported to HEC-RAS. Post-processing options are also available in WMS after running a HEC-RAS simulation.

14.6.1 Creating a schematic and defining roughness values

1. Make the *1D-Hyd Centerline* the active coverage in the Data Tree
2. Select ***River Tools | Map -> 1D Schematic***
3. Switch to the River module
4. Select ***HEC-RAS | Material Properties…***
5. Enter roughness values for Agriculture, Brushland, Bare, River, and Urban as shown in Figure 14-6

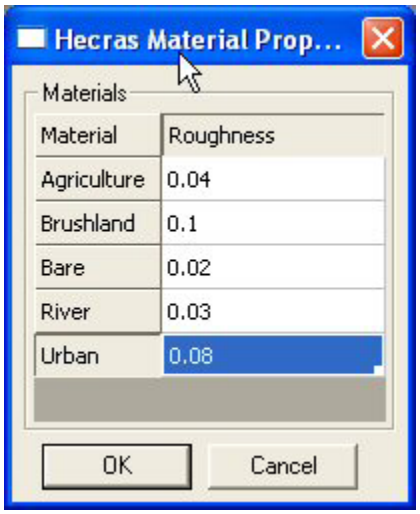
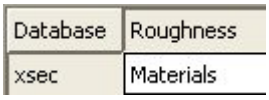


Figure 14-6: Materials roughness values.



6. Select ***HEC-RAS | Model Control…***
7. Select **Materials** to use them to generate roughness values
8. Select *OK*

14.6.2 Running HEC-RAS

1. Select ***HEC-RAS | Export GIS File…***

2. Enter **hecras.geo** and *Save*

This will start HEC-RAS with the geometry file exported from WMS already loaded.

3. Select ***Options | Unit system (US Customary/SI)…***

4. Select *System International (Metric System)*

5. Select *OK*

6. Select *Yes* to set the project units to SI (metric)

7. Select ***Edit | Geometric Data…***

8. Select *OK* on the Schematic Plot Extents error message

9. Select ***View | Set Schematic Plot Extents…***

10. Click the *Set to Computed Extents* button

11. Select *OK* to view the geometric data. It should look similar to the data shown in Figure 14-7

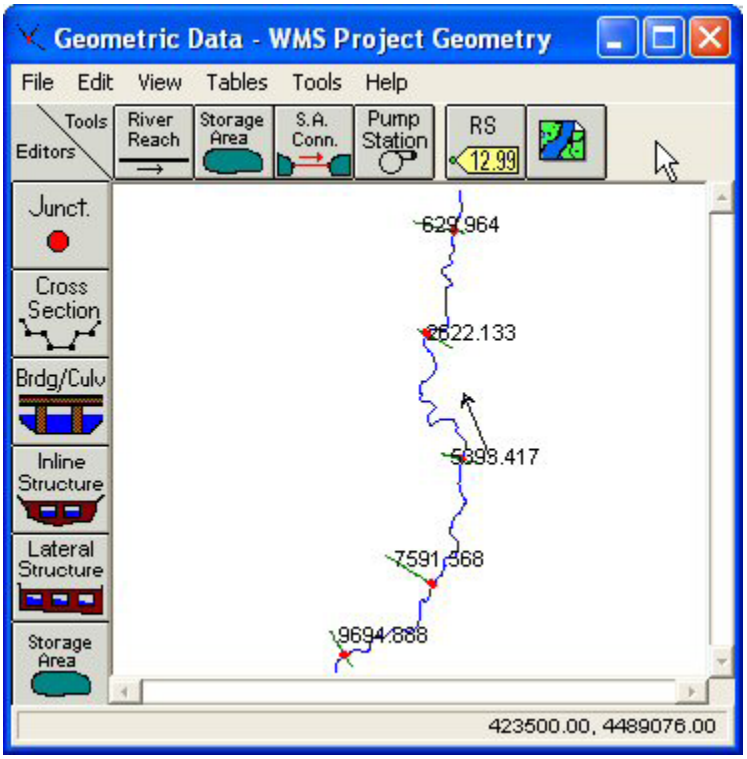
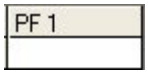


Figure 14-7: Geometric data imported from WMS.

12. Select ***File*** **|** ***Exit Geometry Data Editor*** on the Geometric Data window

13. Select ***Edit*** **|** ***Steady Flow Data…***



14. Enter **80** m^3/s for PF 1 (profile flow rate)

15. Click the *Reach Boundary Conditions* button

16. Click the *Normal Depth* button to enter the *Downstream* boundary condition

17. Enter **0.003** for the downstream slope

18. Select *OK*

19. Highlight the *Upstream* boundary condition and click on the *Normal Depth* button

20. Enter **0.0015** for the upstream slope

21. Select *OK*

22. Select *OK*

23. Click the *Apply Data* button

24. Select ***File*** **|** ***Exit Flow Data Editor*** on the Steady Flow Data window

25. Select ***Run*** **|** ***Steady Flow Analysis…***

26. Change the *Flow Regime* to Mixed

27. Click on the *Compute* button

28. Click the *Close* button

29. Select ***File*** **|** ***Exit*** on the Steady Flow Analysis window

30. Select ***File*** **|** ***Exit*** on the main HEC-RAS window

31. Select *Yes* if prompted to save the project before leaving RAS

14.6.3 Post-processing

1. In WMS select ***HEC-RAS*** **|** ***Read Solution…***

2. Open "*hecras.prj*"

3. Use the Select river reach tool to select the river reach icon displayed on the schematic

4. Select ***HEC-RAS | Plot Solution…*** to view the profile plot generated by HEC-RAS for the river reach

5. Use the Select cross section tool to select one of the cross section icons displayed on the schematic

6. Select ***HEC-RAS | Plot Solution…*** to view the cross section profile plot generated by HEC-RAS

14.7 Floodplain Delineation

HEC-RAS computes a water surface elevation at each cross section. We will interpolate the HEC-RAS results along the cross section and centerline feature arcs in order to improve the floodplain delineation. WMS intersects the water surface elevation data with the background elevation TIN in order to delineate the floodplain.

14.7.1 Interpolating HEC-RAS results

1. Select ***Display | Display Options…***

2. On the River tab toggle *River Hydraulic Schematic* off

3. Select *OK*

4. Switch to the Map module

5. Make the *1D-Hyd Cross Section* coverage active in the Data Tree

6. Make sure that none of the cross section arcs are selected

7. Select ***River Tools | Interpolate Water Surface Elevations…***

8. Choose *Create a data point* **At a specified spacing**

9. Enter **100** for the *Data point spacing*

10. Select *OK*

11. Make the *1D-Hyd Centerline* coverage active in the Data Tree

12. Select ***River Tools | Interpolate Water Surface Elevations…***

13. Select *OK*

14.7.2 Delineating the floodplain

1. Switch to the Terrain Data module
2. Select ***Flood | Delineate…***
3. Enter **500** for *Max search radius*
4. Enter **2** for *Number of stages in a quadrant*
5. Select *OK*
6. Expand the *New tin* folder in the Data Tree
7. Expand the *W.S. Elev-PF 1 (FLOOD)* solution folder in the Data Tree
8. Select ***Display | Contour Options…***
9. For Contour Interval select the *Number of contours* option and enter **25**
10. For Contour Method select *Color fill between contours* and adjust the Transparency
11. Select *OK*

Two data sets were created by delineating the floodplain. *W.S. Elev-PF 1_fd* contains flood depth values and *W.S. Elev-PF 1_wl* contains water surface elevations.

12. Select one of the flood delineation data sets and experiment with viewing the results. You may also want to experiment with adjusting the transparency and turn the background image on as shown in Figure 14-8.

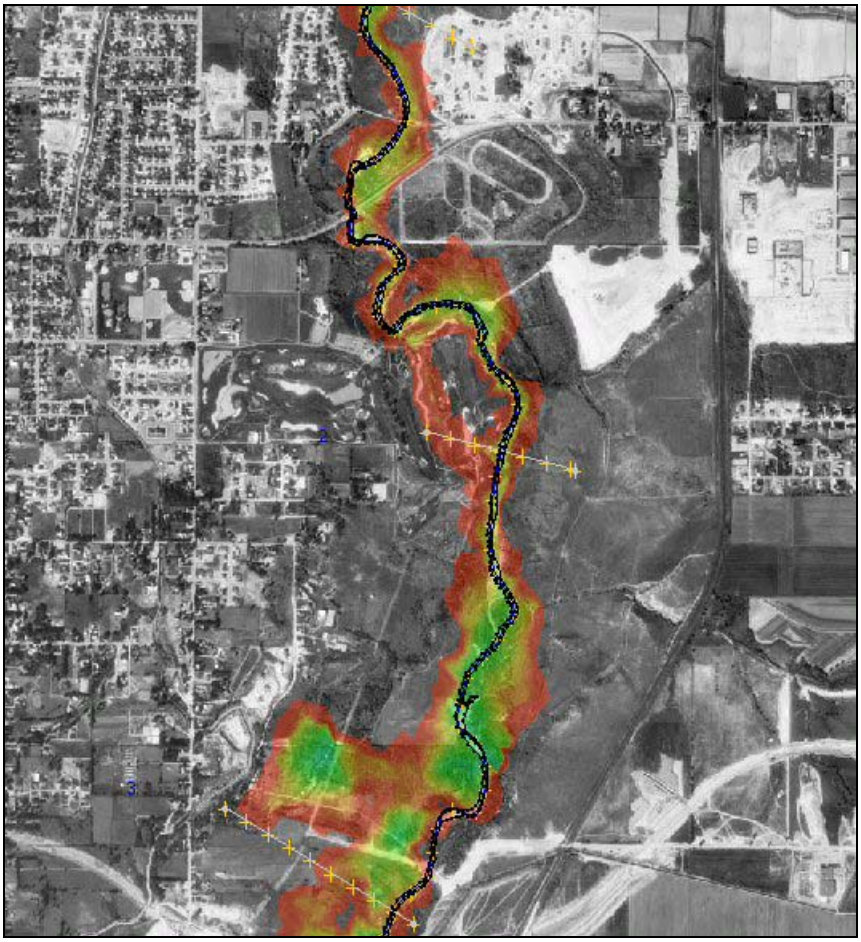


Figure 14-8: Flood depth map.

One important thing to remember is that the HEC-RAS results came from merged cross sections, but the floodplain is delineated on the original TIN that does not include the surveyed channel cross section data. The water surface elevations of the flood are the same, but the flood depths in the channels, where cross sections were merged, may be deeper than shown.

CHAPTER 15

Floodplain Delineation

This tutorial demonstrates how to perform a floodplain delineation with WMS. Before WMS can delineate a floodplain, users must provide it with an elevation TIN (Triangulated Irregular Network) and a scatter point data set with river stage values. TIN elevations might be obtained from survey data, or by converting DEM (Digital Elevation Model) points to TIN vertices. River stage files can be assembled manually, or read in from a HEC-RAS project file.

15.1 Objectives

This tutorial will familiarize you with how to delineate a floodplain based on 1) water surface elevations for a river and 2) a TIN, which represents the topography for the area. The tutorial will teach you how to:

- Experiment with the various Floodplain Delineation options, including input data, search radius, flow path, and quadrants
- Perform floodplain delineations with water surface elevations acquired by the following techniques:
  1. Manually entered in a scatter point file
  2. Approximated with the Channel Calculator in WMS
  3. Computed with HEC-RAS
- Use a Flood Barrier coverage to restrict flood waters

- Generate flood depth, impact, and extent coverages

15.2 Floodplain Delineation Options

You can choose from several different delineation options, which in turn affect how the floodplain is computed. Options include Search Radius, Flow Path, and Quadrants.

The *Search radius* determines how many TIN vertices are taken into account when performing the delineation. The Max search radius is the maximum distance that WMS will "look" from each scatter point to determine the intersection between the water surface and the land elevations. One method for choosing a value for the maximum search radius is to increase the radius until the floodplain extents no longer change.

The *Flow path* option ensures that the interpolated values for the floodplain are hydraulically connected.

The *Quadrants* option makes sure that water level data for interpolation is selected from all directions surrounding the point of interpolation rather than in just one direction (quadrant). In general, it is best to turn on the Quadrants option when computing a floodplain.

Users may want to run several floodplain delineations with varied options in order to see how the floodplain changes. Optimal settings for the delineation options vary with model geometry. For more details on these delineation options, refer to the WMS help file.

To experiment with some of the delineation options, we will open a TIN and a scatter point set. The TIN contains the land surface elevations and the scatter set contains water surface elevations.

1. Select ***File*** | ***Open…***
2. Open "*flood.tin*"

In order to simplify the screen, we will turn off the display of TIN vertices and triangles.

3. Select ***Display*** | ***Display Options…***
4. Choose the *TIN* tab
5. Toggle off the check box for *Unlocked Vertices* (it may already be off)
6. Toggle off the check box for *Triangles*

7. Select *OK*

8. Select ***File | Open…***

9. Open "*samplescatter.wpr*"

10. Expand the *Terrain Data* folder in the Data Tree if necessary

11. Expand the Land TIN in the Data Tree, this way you will be able to see the water surface elevation solution data sets.

12. Select ***Flood | Delineate…***

13. Enter **100** for the *Max search radius*

14. Enter **sr100** for the solution name

15. Select *OK*

16. We will now change the Search radius and re-calculate a floodplain

17. Select ***Flood | Delineate…***

18. Increase the *Max search radius* to **500**

19. Change the solution name to **sr500**

20. Select *OK* to delineate a new floodplain

21. Examine the flood depth solution by expanding the sr100 (FLOOD) solution folder and selecting the sr100_fd data set.

22. Viewing the other flood depth solution reveals a significant difference between the two floodplain delineations. The floodplain extends quite a bit further for the 500 search radius than for the 100 search radius.

23. Delineate two additional floodplains using Max search radii of *1000* and *2000*

24. Expand their solutions in the Data Tree window and select the correspond flood depth data sets (sr1000_fd and sr2000_fd)

You'll notice that there is very little difference between the floodplains delineated with search radius values of *1000* and *2000*. Therefore, we can use a value of *1000* since we have found that increasing the radius doesn't change the delineation appreciably.

Now, we will experiment with using different Flow path values.

25. Select ***Flood | Delineate…***

26. Enter **1000** for the *Max search radius*
27. Toggle on the *Flow path* check box
28. Enter **500** for the *Max flow distance*
29. Rename the solution as **fp500**
30. Select *OK*
31. Delineate two additional floodplains by using *Max flow path* values of **1500** and **3000** and examine their solution sets.

Once again, there is little variation in these last two delineations. Therefore, we might leave our Flow path value at 1500. As a note, when the TIN includes an area of unusually high elevations near the river (such as a levee), it is a good idea to use the Flow paths option.

Experiment further with the display options until you get a feel for how they change the floodplain delineation.

15.3 Creating a Scatter Point File

A scatter file, similar to the one you opened in the previous section, can be created with any text editor or spreadsheet application. The required input for the file are (x, y) coordinates for each data point, and a corresponding data set (in this case, the data set would be water surface elevation values for each coordinate). Each coordinate may be associated with more than one data value. Figure 1-5 shows the file format for a 2D scatter point file:

ID	X	Y	"dataset 1"	"dataset 2"	…
1	2343	32322	34.3	45.7	…
2	2348	32318	33.9	45.4	…
3	2350	32316	33.5	45.0	…
etc					

Figure 15-1: File format for 2D scatter point file

1. Select ***File*** | ***New***
2. Select *No* when asked if you want to save changes

For this tutorial, the scatter file has been created for you. We will open it with a text editor to view how it is set up:

3. Select ***File*** | ***Edit File…***
4. Open "*wse.txt*"

5. Select *OK* to open the file with *Notepad*

The datasets for the water surface elevations are named "WSE1" and "WSE2." The WSE1 data set might represent current water levels, and WSE2 might represent expected levels given future developments, such as a proposed levee.

6. Close *Notepad*

15.3.1 Open the Scatter Point Data

1. Select ***Flood*** | ***Read Stage File…***
2. Open "*wse.txt*"
3. Select ***File*** | ***Open…***
4. Open "*flood.tin*"

15.3.2 Delineate the Floodplain

1. Select ***Flood*** | ***Delineate…***
2. Choose *WSE1* from the *Select stage data set* list
3. Change the *Max search radius* to **1500**
4. Toggle off the *Flow path check* box
5. Change the *Number of stages in a quadrant* to **3**
6. Select *OK*
7. This will begin the delineation process for the first set of water surface elevations. When WMS finishes, delineate a new floodplain based on the second set of elevations:
8. Select ***Flood*** | ***Delineate…***
9. Choose *WSE2* from the *Select stage data set* list
10. Select *OK*

15.3.3 Creating a Flood Impact Map

WMS can use two separate floodplain delineations to generate a Flood Impact coverage.

A Flood Impact coverage shows the difference between two flood depth or water level sets. The differences are divided into ranges or classes. Using the floodplains delineated in the previous steps, you will create a Flood Impact coverage in order to investigate the difference in flooding with and without the proposed levee.

1. Select ***Flood*** | ***Conversion*** | ***Flood -> Impact Map…***
2. Choose *WSE1_fd* in the *Original data set* combo box
3. Choose *WSE2_fd* in the *Modified data set* combo box
4. Toggle on the *Create "no impact" polygons* check box
5. Select *OK* to accept the default classes, ranges, attributes, and names
6. Select *Yes* to create the coverage
7. This new data set is calculated as *WSE1_fd – WSE2_fd* indicating that all values in the second data set were subtracted from their corresponding values in the first data set.
8. Switch to the *Map* module
9. Choose the *Select Feature Polygon* tool
10. Double-click on any one of the polygons
11. The dialog that opens shows the amount of change from the original data set to the modified data set, as well as the impact class ID and name.

Besides creating a tabular data file as described above, scatter point sets can also be created interactively with the tools in WMS, or opened from a HEC-RAS solution file. The following sections demonstrate these two methods for constructing scatter point data to be used in a floodplain delineation.

15.4 Creating Scatter Points with the Channel Calculator

Users can interactively create scatter points in the 2D Scatter Point module and enter a water surface elevation for each point. This section will demonstrate this method. To begin, we will clear all data from WMS and begin afresh.

1. Select ***File*** | ***New***
2. Select *No* when asked if you want to save your changes

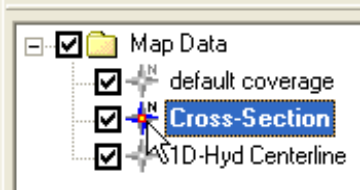
WMS has a tool that allows us to interactively create scatter points and assign a data value to each point. The values for water surface elevations might be obtained by digitizing from a background image, or from another source. You can also use a cross section coverage in conjunction with the channel calculator to create scatter points with calculated depths along a river centerline.

For this example, we will use the Channel Calculator to compute water depths for various cross-sections in our channels and create a scatter point at each cross-section arc. The water surface elevation will be equal to the water depth computed by the Channel Calculator plus the ground elevation (given by the TIN).

We will begin by opening a MAP file containing a Cross Section coverage. This coverage contains several cross-section arcs along our stream reaches.

15.4.1 Open the Cross-section Arcs and River Centerlines

1. Select ***File*** | ***Open…***
2. Open the file named *"flood.tin"*
3. Select ***File*** | ***Open…***
4. Open "*cross_section.map*"
5. Make sure you are in the *Map* module
6. Right click on the *Coverages* folder in the Data Tree and select *New Coverage* from the pop-up menu
7. Change the Coverage type to *1D-Hyd Centerline*
8. Select *OK*
9. Select ***File*** | ***Open…***
10. Open "*reaches.shp*"
11. Select *OK*



12. Select the *Cross-Section* coverage from the Data Tree window so that it will be the active coverage.

15.4.2 Define Manning's Roughness Values for Cross Sections

13. Choose the *Select Feature Arc* tool

14. Select ***Feature Objects*** | ***Attributes…***

This dialog displays the cross-section elevations and allows you to edit the cross-section stations and/or elevations, and to assign Manning's roughness values. We will increase the Z scale to better distinguish the elevations:

15. Change the Z scale to *25:1*

16. Choose *Cross section 1* from the list of Cross Sections

17. Select the *Edit multiple n values* button

18. Enter a value of **0.03**

19. Select *OK*

20. Select *OK* when asked if you want to change all Manning's n values

21. Repeat the above steps to assign Manning's n values to the remaining cross-sections as shown in the following table

Cross Section Name	Manning's n
Cross section 2	.023
Cross section 3	.033
Cross section 4	.026
Cross section 5	.031
Cross section 6	.027

1. Select *OK* to close the Attributes dialog when you have finished defining all Manning's n values

15.4.3 Using the Channel Calculator to Compute Depths

The Channel Calculator is a good tool for approximating channel flows or flow depths. Given a flow rate, the Calculator can compute a flow depth, and vice versa. As we calculate flow depths, we need to jot down the depth values so we can recall them later when we create the 2D scatter points.

1. Switch to the *Hydrologic Tree* module

2. Select ***Calculators*** | ***Channels…***

3. Change the Channel Type to *Cross Section*

4. Make sure that *Cross section 1* is selected from the Cross section list

5. Change the Units to *Metric*

6. Enter a value of **0.002** for *Longitudinal slope.* This is an estimate for the ground slope in the vicinity of Cross section 1.

7. Enter a value of **450** for the flow

8. Select the *Calculate* button

9. Select the *Create Stage Point* button

10. Choose *Cross section 2* from the *Cross Section* list

11. Repeat the previous steps to compute Depths and create a stage point for this and all remaining cross-sections. The following table provides Flow and Longitudinal slope values for use with each cross-section.

Cross Section	Flow (cms)	Slope
Cross section 1	450	.0020
Cross section 2	450	. 0015
Cross section 3	300	. 0019
Cross section 4	300	. 0006
Cross section 5	150	. 0087
Cross section 6	150	. 0037

The Channel Calculator can also be used to generate a rating curve

1. Select the *Compute Curves…* button

The default is to create a rating curve for the entered flow vs. depth, but you can also create curves for all of the other options listed. Further, you can enter a depth in the Channel Calculator and compute Depth on the Y Axis vs. any of the listed options (Flow would replace the Depth option for the X Axis)

2. Select *OK*

You can right-click in the rating curve plot window to export the data to a spreadsheet if you want.

3. When you are done viewing the rating curve plot, close the plot window by selecting the X in the upper right corner of the window

4. Select *OK* to close the Channel Calculator

15.4.4 Interpolating Stages along the Centerline

You should now have a scatter point created with a computed water surface elevation at each cross section where it intersects the centerline as shown in Figure 15-2. In order for the flood delineation to work better we want to interpolate values along the centerline.

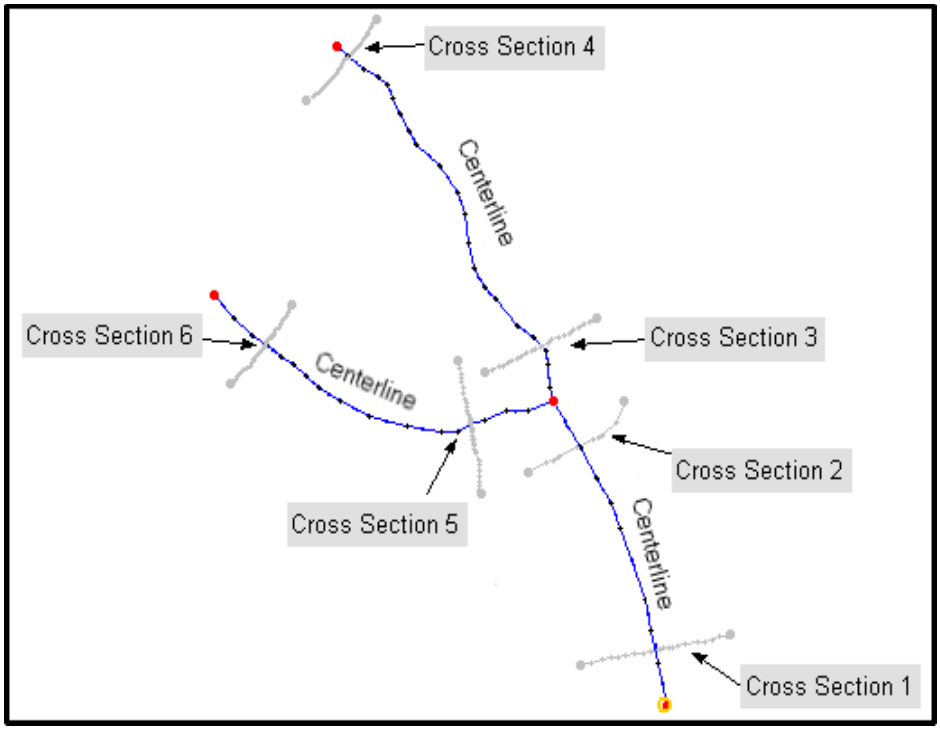


Figure 15-2: Labels for each cross section.

1. Switch to the *Map* module

2. Make sure the *1D-Hyd Centerline* coverage is active in the Data Tree (this should also activate the River Tools menu in the Model drop-down list).

3. Select ***River Tools | Interpolate Water Surface Elevations…***

4. Set the Create a data point field to *At a specified spacing*

5. Enter **30** for the spacing

You should see many scatter points added along the centerline arcs, resulting in a view similar to that in Figure 15-3 (the symbol for scatter points may be different).

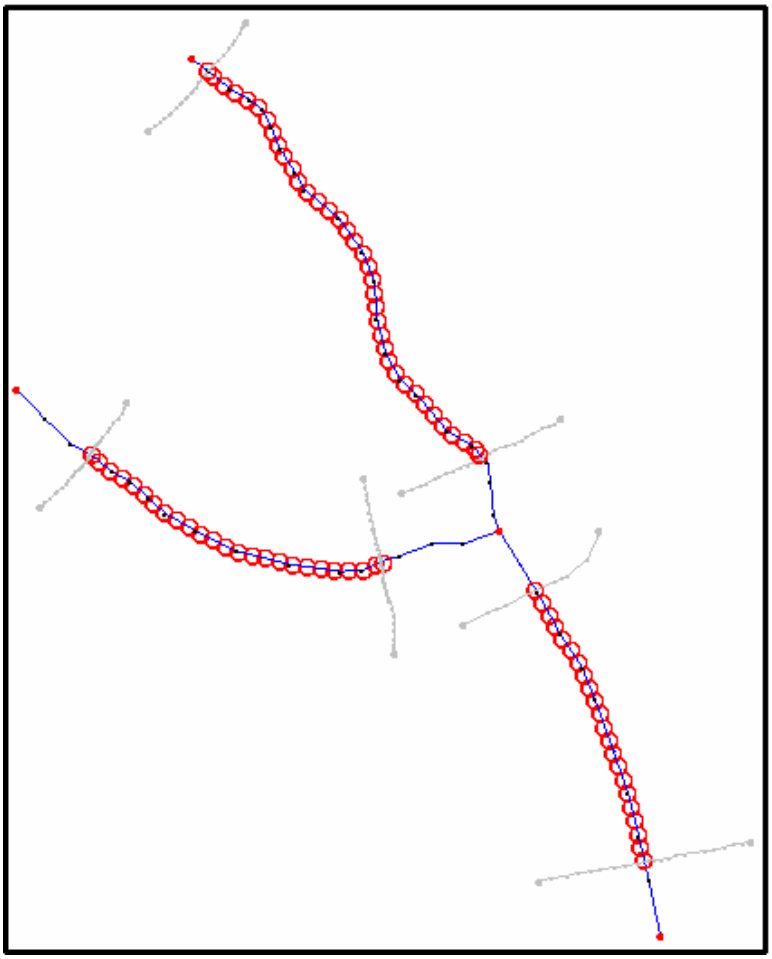


Figure 15-3: View of interpolated scatter points.

If your scatter points didn't interpolate as shown above, the most probable reason is that the original scatter points were not placed close enough to the Centerline arc. In order to continue, delete the scatter point set and either try creating the points again, or open the sample scatter point set contained in "*samplescatter.wpr*".

15.4.5 Delineate the Floodplain

1. Switch to the *Terrain Data* module
2. Select ***Flood*** | ***Delineate…***
3. Change the *Max search radius* to **1000**
4. Toggle on the *Flow path* option and enter a *Max flow path* value of **1000**
5. Select *OK*

6. Turn on the display of the Flood plain contours to view the results

We have finished delineating a floodplain based on the water elevations that we calculated on our own. The next section will describe how to open a set of scatter points that were entered into a text file.

15.5 Delineation from HEC-RAS Data

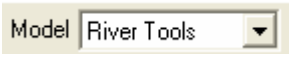
15.5.1 Reading the HEC-RAS solution

In this section, we will use water surface elevations computed as a result of a separate tutorial that builds a HEC-RAS project. First, we will read a WMS project file that contains a TIN and then import a HEC-RAS solution from a model developed from the TIN:

1. Select ***File*** | ***New***
2. Select *No* when asked if you want to save changes
3. Select the *Terrain Data* module
4. Select ***File*** | ***Open…***
5. Open "*flood.wpr*"

6. To simplify the display, we will hide some elements of the model
7. Expand the *Terrain Data* folder from the Data Tree if necessary
8. From the Data Tree, hide the TIN by toggling off its check box
9. Select ***Display*** | ***Display Options…***
10. Choose the *River* tab
11. Toggle off the *River Hydraulic Schematic* check box
12. Select *OK*
13. Switch to the *River* module
14. Select ***HEC-RAS*** | ***Read Solution…***
15. Open "*hecrun.prj*"

The HEC-RAS solution is read in as a set of scatter points, with one water surface elevation for each cross-section. The floodplain delineation interpolation will work much better if there are more points than the sparsely spaced points that are part of the solution. To increase the density of the scatter points, we will interpolate between existing points to create additional points along the centerline and cross-section arcs. Since the water surface is assumed constant along a cross section and varies linearly between this does not violate any of the modeling assumptions.



1. Switch to the *Map* module
2. Expand the *Map Data* folder in the Data Tree, if necessary
3. Toggle off the check box for the Materials coverage
4. Activate the *1D-Hyd Centerline* coverage in the Data Tree
5. If needed, choose *River Tools* from the Model drop-down list
6. Select ***River Tools | Interpolate Water Surface Elevations …***
7. Choose *At a specified spacing* for the Create a data point option
8. Enter **60** for the Data point spacing
9. Select *OK*
10. Select the *1D-Hyd Cross Section* coverage from the Data Tree
11. Select ***River Tools | Interpolate Water Surface Elevations …***
12. Select *OK* to interpolate with the options set as before

You should now see that your screen is more densely populated with scatter points. Note that along the centerline arcs, the scatter points have been interpolated in a linear fashion, while along cross-section arcs, the points that were added have the same data value as the original point.

15.5.2 Using a Flood Barrier Coverage

WMS allows users to "confine" a delineation from the given elevation data by creating a flood barrier coverage. Arcs representing ridges or levees (existing or proposed) may be created in the model, and these in turn alter the floodplain delineation by restricting interpolation of the floodplain so that values on the "dry" side of the levee are not interpolated. In this tutorial, a map file of arcs representing a proposed levee will be used to demonstrate the effects of incorporating a flood barrier coverage.

1. Select ***File*** | ***Open…***

2. Open the file named "*levee.map"*

Two new coverages are now added to the Map Data folder in the Data Tree. The coverage entitled *proposed levee* is a Flood Barrier coverage and *breakline* is a General coverage.

3. Verify that *proposed levee* is the active coverage in the Data Tree

4. Select the *Zoom* tool

5. Zoom in around the proposed levee as shown in Figure 15-4. The proposed levee is located along the west bank of the tributary stream.

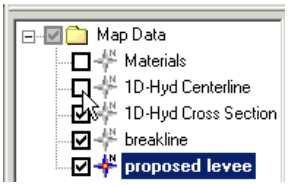
Figure 15-4: Zoom in on the arc representing the levee

Contouring of floodplain data sets in WMS is based on TIN vertices. Therefore, it is important to make any necessary changes to the TIN before performing floodplain delineation, especially if comparisons are to be made between different scenarios.

The flood extent is contoured to midpoints on triangle edges between flooded and dry areas.It is therefore recommended that a breakline be created on the "river side" of the flood barrier coverage in order to improve the visualization of the delineated floodplain. This breakline and the flood barrier are then forced into the TIN, effectively confining the flood contours between the two.

You will now force the flood barrier and breakline arcs into the TIN. The breakline is located immediately to the east of the flood barrier (you probably can't see it unless you zoom in closely around the flood barrier arc).

To better distinguish the proposed levee and breakline arcs, hide the 1D-Hyd Centerline coverage.



1. Toggle off the check box for *1D-Hyd Centerline* in the Data Tree
2. Choose the *Select Feature Arc* tool
3. Select the flood barrier arc
4. Select ***Feature Objects | Arcs -> Breaklines***
5. When prompted, choose *Use all arcs as breaklines*
6. Select *OK*
7. At the next prompt, choose *Interpolate Z values from existing TIN*
8. Select *OK*
9. Activate the *breakline* coverage by selecting it from the Data Tree
10. Click on the *Select Feature Arc* tool
11. Select the breakline arc
12. Select ***Feature Objects | Arcs -> Breaklines***
13. When prompted, choose *Use all arcs as breaklines*
14. Select *OK*
15. At the next prompt, choose *Interpolate Z values from existing TIN*
16. Select *OK*

15.5.3 Delineating the Floodplain

Now that we have added the breaklines to our TIN, we are ready to delineate the flood plain:

1. Switch to the *Terrain Data* module
2. Select ***Flood | Delineate…***

3. Choose the *User defined flood barrier coverage* option
4. Make sure the *Search radius* option box is toggled on and enter **1000** for the *Max search radius*
5. Enter **500** for the *Max flow distance*
6. Make sure the *Quadrants* check box is toggled on
7. Enter **4** for the *Number of stages in a quadrant*
8. We will accept the default solution and data set names
9. Select *OK*

It may take a minute or two for WMS to compute the floodplain delineation

10. From the Data Tree, toggle-on the TIN's check box to show it again
11. Select the Frame Image macro
12. Expand the folder named *Land*
13. Expand the folder named *W.S. Elev-PF 1 (FLOOD)* in the Data Tree
14. Select the data set named *W.S. Elev-PF 1_fd*
15. Your screen should appear similar to Figure 15-5

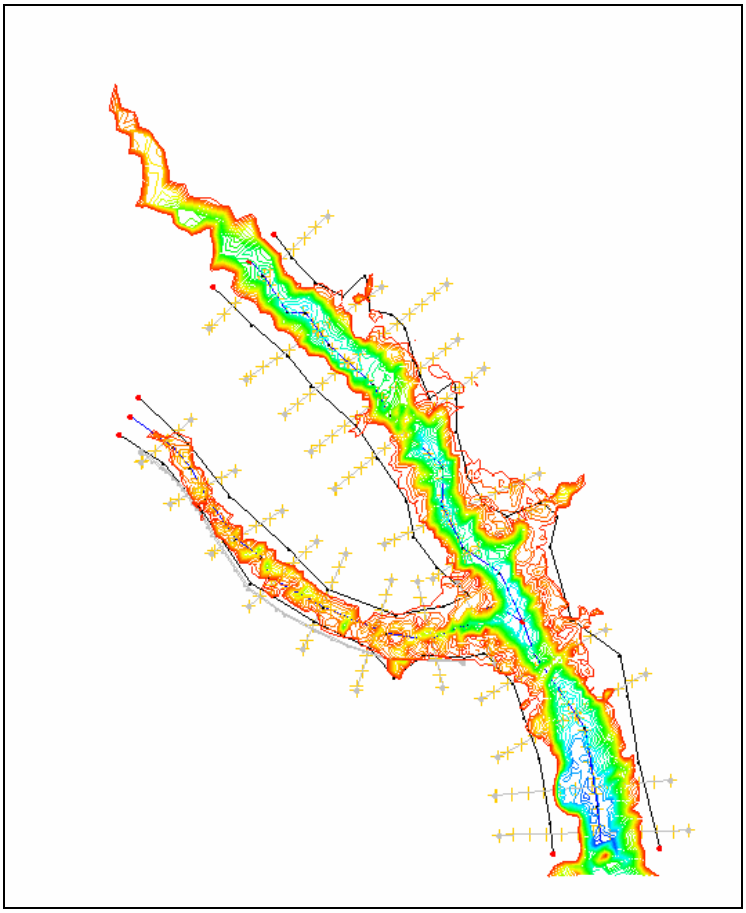


Figure 15-5: Plot of floodplain depths

These contours correspond to the water depths in the floodplain area. To view the water surface elevation data set:

16. Select the data set named *W.S. Elev-PF 1_wl* from the Data Tree

15.6 Creating a Flood Extent Coverage

Flood depth and water level information are stored with the TIN, but WMS allows for the creation of feature objects from this data. In floodplain delineation, it may be useful to create a flood extent coverage. This coverage defines the boundary of the flood and may be exported for use in GIS applications. To create the flood extent coverage:

1. Select ***Flood* | *Conversion* | *Flood -> Extent Coverage…***

2. Select *W.S. Elev-PF 1_fd* from the *Select Flood Depth Data Set* list

3. Select *OK* (we will accept the default inundation limit and coverage name)

The flood extent boundary is converted to feature lines and WMS will try to build a polygon enclosing the flooded area. However in this case it reaches the boundary of the TIN and so a complete polygon is not available. You could close the polygon manually be creating an arc to along the TIN boundary and then choosing to Build Polygons.

15.7 Creating a Flood Depth Coverage

The flood extent coverage essentially divides the watershed area into two parts: flooded and not flooded. However, it is often necessary to know not only if an area is flooded but also how badly it is flooded.

It is common to divide the flooded area into zones, each with a depth range. In WMS, these zones are created by making a flood depth coverage. To create a flood depth coverage:

1. Select ***Flood | Conversion | Flood -> Depth Map***
2. Select *W.S. Elev-PF 1_fd* from the *Select Flood Depth Data Set* combo box
3. Note the ranges and attributes of the five "zones" or "flood classes"
4. Select *OK*
5. Select the *Zoom* tool
6. Zoom in to view the bottom portion of the main channel
7. Switch to the *Map* module
8. Click on the *Select Feature Polygon* tool
9. Double-click inside a few of the polygons that have been created. This will bring up the flood extent attributes, which include the average flood depth for the zone.

15.8 Conclusions

After completing this tutorial, you should be familiar with using WMS to perform floodplain delineations in conjunction with:

- A flood barrier coverage
- Water surface elevations computed with HEC-RAS

- User-defined water surface elevations, created either interactively, or opened with a 2D scatter point file

- Post-processing in the form of creating Flood Extent, Flood Depth, and Flood Impact maps

CHAPTER 16

Stochastic Modeling Using HEC-1 and HEC-RAS

This tutorial demonstrates how to run a stochastic model within WMS. Since a certain degree of uncertainty usually exists in the selection of input parameters, running a stochastic model helps us explore multiple model solutions for differing input values. Users can specify a range of values to be used in the stochastic model.

16.1 Objectives

In this tutorial, you will learn the basics of setting up a stochastic model that will run the HEC-1, HEC-RAS, and Floodplain Delineation models simultaneously.

1. Assign key values to parameters
2. Define the Stochastic Model characteristics
3. Run the Model
4. View the solution means and probabilities

16.2 Opening the HEC-1 and HEC-RAS Models

Working HEC-1 and HEC-RAS models have been previously prepared for use in this tutorial. We will use the Stochastic modeling features of WMS to create a probabilistic map of floodplain boundaries. You may choose to review the development of these models in their separate tutorials.

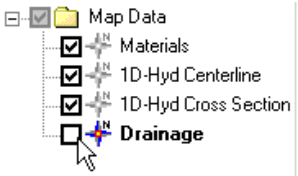
There is generally a high degree of uncertainty associated with hydrologic modeling parameters such as the SCS Curve Number and rainfall depth. Using WMS you can vary these parameters stochastically in order to analyze a more complete parameter space and then use the results to reflect the known uncertainty in a flood plain delineation. The Stochastic Simulation will run a specified number of simulations using randomly generated values of basin Curve Numbers and basin precipitation for each simulation. Results from the HEC-1 run are then passed to a developed HEC-RAS simulation, and finally the results from HEC-RAS are used to compute a series of flood plain boundaries from which a probability of flooding, based on model parameter uncertainty can be computed.

1. Select ***File*** | ***Open…***
2. Open "*run1.wpr*"

Run1.wpr is a WMS project file that references all of the prepared input data for the HEC-1 and HEC-RAS models. Each simulation has been previously tested to ensure that they can be run successfully.

16.2.1 Preparing the HEC-RAS model

1. To begin, we will zoom in on the HEC-RAS model domain.
2. Switch to the *Tree* module
3.

Hide all basins and outlets by un-checking the box next to the Hydrologic Tree Data folder in the Data Tree
4. Switch to the *Map* module
5. 

Expand the Map Data folder in the Data Tree by clicking its + icon
6. Hide the coverage named *Drainage* by un-checking its visibility box in the Data Tree
7. Select the *Frame* macro
8. Activate the *1D-Hyd-Centerline* coverage by single-clicking on its name in the Data Tree window

9. Select ***River Tools | Map -> 1D Schematic***. This step initializes the HEC-RAS river scheme in preparation for reading in the HEC-RAS solution.

10. Switch to the *River* module

11. Select ***HEC-RAS | Read Solution…***

12. Open "*hecrun1.prj*"

When we read in a HEC-RAS project file in which the water surface profiles have been calculated, a scatter point is placed at the thalweg location of each cross-section. Each scatter point contains the value of the water surface elevation at that point in the river reach. To create additional scatter points for interpolation purposes, we need to interpolate more scatter points along both the center-line and cross-section arcs.

13. Switch to the *Map* module

14. Make sure the *1D-Hyd-Centerline* coverage is active in the Data Tree

15. Select ***River Tools | Interpolate Water Surface Elevations…***

16. Set the Create a data point field to *At a specified spacing*

17. Enter **60** for the Data point spacing

18. Select *OK*

19. Activate the *1D-Hyd-Cross Section* coverage from the Data Tree

20. Select ***River Tools | Interpolate Water Surface Elevations…***

21. Select *OK*

This same data point spacing (60) will be used to interpolate each HEC-RAS solution along the reach and the cross sections when running the stochastic simulations.

16.2.2 Preparing the HEC-1 model

1. Show the *Drainage* coverage by checking its box in the Data Tree

2. Switch to the *Tree* module

3. Show the contents of the Hydrologic Tree Data folder by toggling on its check box

4. Select the *Frame* macro

We will now assign key values to the parameters that we wish to randomly vary during the Stochastic run. Key values are negative integers that we assign to a parameter in lieu of the actual value. By entering a negative value for precipitation, for example, WMS replaces the precipitation value with a feasible precipitation value at run time. This precipitation value changes for each new run. Each generated value is based on a normal distribution, and can be controlled by specifying minimum and maximum allowable values.

5. Choose the *Select Basin* tool
6. Multi-select the basins labeled 2B and 3B by holding the <Shift> key as you single-click on each basin icon
7. Select ***HEC-1 | Edit Parameters…***
8. Select the *Precipitation* button
9. Enter a value of **-1** in the Average precipitation field
10. Select *OK*
11. Select the *Loss Method* button
12. Enter a value of **-2** in the CRVNBR field
13. Select *OK*
14. Select *Done*
15. Multi-select the basins labeled 4B and 1B
16. Use the method shown in the previous steps to assign an Average precipitation value of **-1** and a CRVNBR value of **-3** to these basins

16.3 Running the Stochastic Model

For this tutorial, we have assigned key values only to HEC-1 parameters, namely precipitation and Curve Number. However, WMS also allows key values to be assigned to Manning's N values for the HEC-RAS model, and Search Radius for the Floodplain Delineation model.

16.3.1 Setting Up the Model

1. Select ***HEC-1 | Stochastic Modeling…***

Users can choose either a Monte Carlo or Latin Hypercube approach to generating values for the stochastic model. We will use the Monte Carlo approach for this exercise. When using the Monte Carlo method, we must specify the number of simulations that we want to run. With the Latin Hypercube approach, instead of specifying a number of simulations, we enter a number of segments for each variable. The total number of simulations for a Latin hypercube run is equal to the product of the number of segments for each variable. If we have 3 variables, with 3 segments each, then a Latin Hypercube model will run 27 times. Users also specify maximum, minimum, and starting values for each stochastic variable.

2. Select the *Monte Carlo* option for the Simulation type

3. Enter **5** for the Number of simulations

4. We specify 5 simulations for this tutorial in the interest of time. Remember, though, that statistical studies indicate that in order for the generated values to resemble a normal distribution, one should run at least 30 simulations.

5. Choose *HEC-1* from the *Selected model* list box

6. Toggle on the *Define stochastic model* check box

7. Under the Filename parameters section, select the *Base filename:* button

8. Browse to the folder in which the HEC-RAS project file ("*hecrun1.prj*") is located and enter **hecrun1** as the File name

9. Select *Save*

10. Select the *Add variable* button 3 times to add 3 variables to the table

11. Refer to Figure 16-1 and enter the values shown for each stochastic variable

	Used	Name	Key	Type	Sta...	Min ...	Ma...	Stand...	Distribution	Log
1	☑	HEC1_1	-1	Precipitat...	6.0	3.0	9.0	2.5	Normal	☐
2	☑	HEC1_2	-2	Curve Nu...	68.0	62.0	72.0	4.0	Normal	☐
3	☑	HEC1_3	-3	Curve Nu...	76.0	70.0	80.0	4.0	Normal	☐

Figure 16-1: Values for the Stochastic Variables table

12. Now, we will turn on stochastic modeling for the HEC-RAS and Floodplain models:

13. Change the *Selected model* list box to *HEC-RAS*

14. Toggle on the *Define stochastic model* check box

15. Select the *Base filename* button and browse to the same path as "*hecrun1.prj*"
16. Enter **hecrun1** as the File name and select *Save*
17. Change the *Selected model* list box to *Floodplain*
18. Toggle on the *Define stochastic model* check box
19. Select the *Base filename* button and browse to the same path as "*hecrun1.prj*"
20. Enter **hecrun1** as the File name and select *Save*
21. Select *OK* to exit the Stochastic Run Parameters dialog
22. Select ***File | Save*** to save your changes

16.3.2 Running the Model

The last step in setting up the stochastic simulation is linking the output hydrographs (in this case it is a steady state simulation and so only the peak flow is used) from the HEC-1 model to the HEC-RAS input boundary condition for the appropriate rivers and reaches.

1. Select ***HEC-1 | Run Stochastic Model…***
2. Choose *2B Basin Hydrograph* from the Select a hydrograph combo box
3. Choose *Leith River* from the Select a river combo box
4. Choose *Upper Branch* from the Select a reach combo box
5. Choose *2648.528* from the Select a cross-section list box
6. Select the *Assign Peak Flow BC* button
7. Repeat the previous steps to link the hydrographs and cross-sections shown in Table 16-1:

Table 16-1: Assigning additional hydrographs to cross-sections

Hydrograph	River	Reach	Cross-section
3B	Left Fork	Branch	1703.086
5C	Leith River	Lower Branch	759.138
2C	Leith River	Lower Branch	72.889

Before we use the Stochastic Model to run Floodplain Delineation over and over again, we need to change some of the delineation options. To set these options:

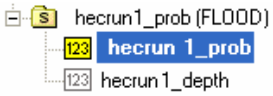
8. Select the *Floodplain Delineation Options* button.
9. Change the Max search radius to **1500**
10. Make sure the *Quadrants* check box is toggled on
11. Enter **3** for the Number of stages in a quadrant
12. Select *OK*
13. Select *OK* to run the Stochastic simulations

Progress for the simulations will be displayed in the Stochastic dialog. You will be able to see the values selected for the CN and precipitation for each run as well as the diagnostic information for the models. It is possible to abort prematurely by selecting the *Abort* button.

16.4 Viewing the Results

All solutions (HEC-1 hydrographs, HEC-RAS river stages, and Floodplain delineations) are read into WMS. Furthermore, at the completion of the stochastic run, WMS computes a dataset containing the probabilities that flooding will occur at various locations within the original TIN. This is computed by assigning to each vertex in the TIN whether it was flooded or not. At the completion of all runs contours showing 0-100 percent probability of flooding can be viewed. 100% probability would mean that flooding occurred for the entire model, 50% probability would be flooding for half of the simulations, etc. To view the results:

1. Select *Close* to exit the Stochastic HEC-1 wrapper
2. Switch to the *Terrain Data* module
3. Select ***Display | Display Options…***
4. Choose the *TIN* tab
5. Toggle on the *TIN Contours* check box



6. Select *OK*
7. From the Data Tree, expand *New Tin* in order to see its subfolders
8. Expand the folder named *hecrun1_prob(FLOOD)*
9. Select the dataset named *hecrun1_prob*
10. By contouring this dataset, we see the probabilities that the floodplain will arrive at different locations.
11. Now, select the dataset named *hecrun1_depth*

This dataset contains the average depths from all of the stochastic simulations.

CHAPTER 17

Simplified Dam Break

Simplified Dam Break (SMPDBK) is a model that does just what its name says—it models dam failures using simplified methods. One alternative to using SMPDBK is to use sophisticated dam break models such as the National Weather Service's (NWS) DAMBRK model. These models require extensive data, time, and computing power. When these data or resources are not available, SMPDBK can be used to create a "quick and dirty" solution to the flood depths downstream of a dam failure. By combining the SMPDBK results with the floodplain delineation and display capabilities of WMS, you can create a good picture of the aerial extents of this flood.

17.1 Preparing the Model

17.1.1 Running TOPAZ

In this section, you will read the DEM and run TOPAZ to compute the flow directions and flow accumulations on the DEM. The purpose of doing this is to obtain a stream arc that represents the centerline of the stream downstream from the dam. This stream arc will be used in a 1D Hydraulic Centerline coverage to create the geometry for the SMPDBK model in WMS.

1. Select ***File*** | ***Open…***
2. Open "*smpdbk.gdm"*

3. Switch to the Drainage Module
4. Select ***DEM | Compute Topaz Flow Data …***
5. Select *OK*
6. Select *Current Coordinates…* from the *Units* dialog.
7. In the *Current Coordinates* dialog, select *UTM NAD 83 (US)* for the horizontal system, *Meters* for the horizontal units, *12—114W to 108W* for the UTM Zone, *NAVD 88 (US)* for the vertical system, and *Meters* for the vertical units.
8. Select *OK.*
9. Select *OK*
10. Once TOPAZ finishes running (you may have to wait a few seconds to a minute or so), choose *Close* to close the TOPAZ window

You should now see a network of streams on top of your DEM. TOPAZ computes flow directions for individual DEM cells and creates streams based on these directions. You can change the flow accumulation threshold so that smaller or larger streams show up.

11. Select ***Display | Display Options …***
12. On the DEM tab, change the Min Accumulation for Display to 5.0.
13. Select *OK*

14. Right click on "DEM" in the *Terrain Data* folder in the Data Tree and select *Coordinate Conversion*
15. Click the checkbox next to *Edit project coordinate system*
16. Change the horizontal and vertical units to *U.S. Survey Feet*.
17. Select *OK*

17.1.2 Creating Outlets and Streams

The next step in creating a SMPDBK model is to convert the computed TOPAZ flow data to a stream arc. This arc can then be used as the stream centerline in the SMPDBK model.

1. Choose the *Create Outlet Point* tool

2. Create an outlet on the river in the **lower left corner** of the DEM, as seen in Figure 17-1. Be sure to click close enough to the river so the outlet snaps to the flow accumulation cell on the stream. The dam is located in the upper right corner of the DEM.

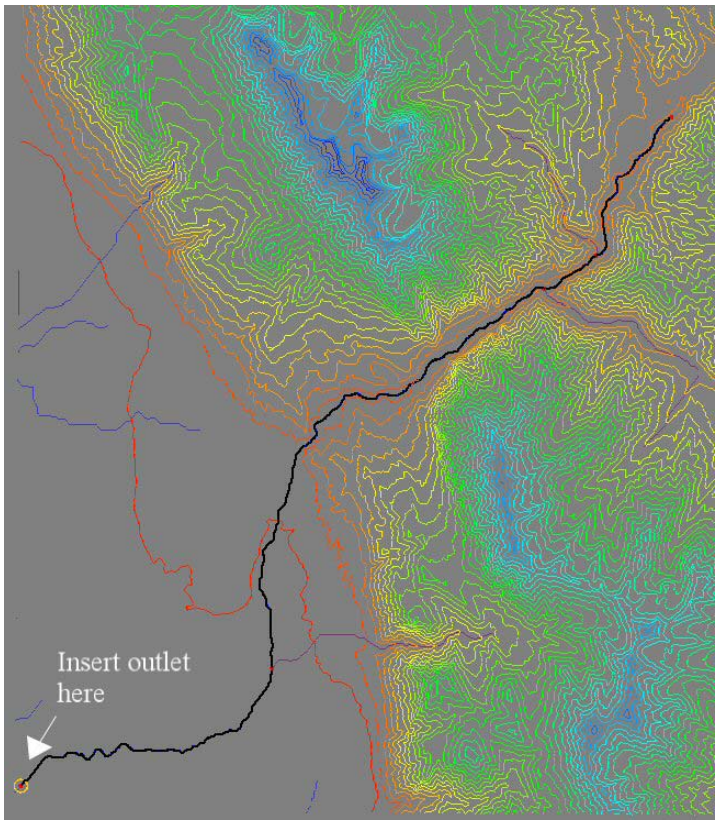


Figure 17-1: New outlet point.

3. Select ***DEM | DEM -> Stream Arcs …***

4. Select *OK*

5. Choose the *Select Feature Arc* tool

6. While holding down on the <SHIFT> key, select the three stream arcs that branch off of the main arc.

7. Press *Delete*

8. Select *OK*

You have now isolated the main stream arc. Your screen should look like Figure 17-2.

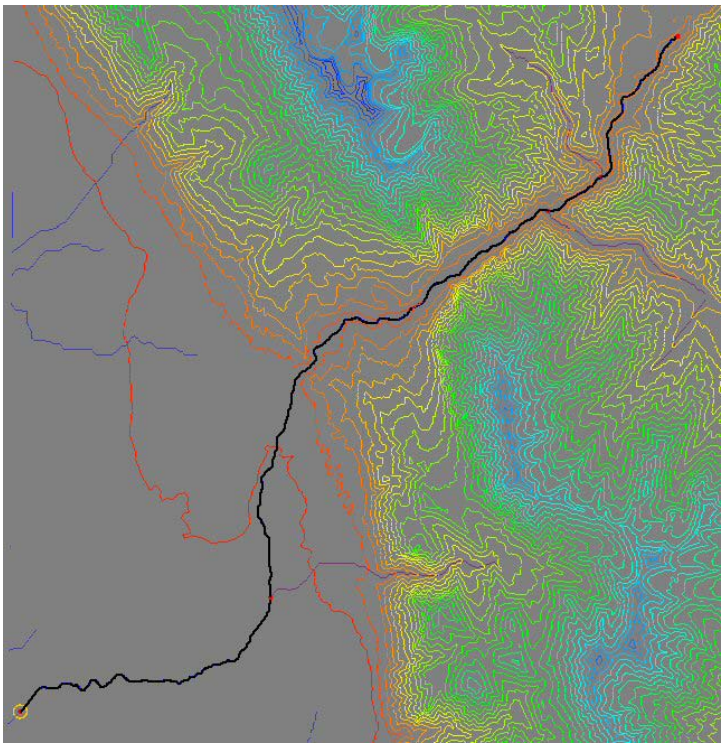


Figure 17-2: Main stream arc.

17.1.3 Creating 1D-Hydraulic Coverages

The next step is to create arcs representing the stream centerline (in a 1D Hydraulic Centerline coverage) and to create cross section arcs along this centerline (in a 1D Hydraulic Cross Section coverage).

1. Switch to the *Map* module
2. Choose the *Select Feature Point/Node* tool
3. Drag a box around the entire stream arc. Five nodes should be selected.

4. Select ***Feature Objects*** | ***Vertex <-> Node.*** This will convert all the selected nodes to vertices, turning the stream centerline into a single arc.

5. Select the stream centerline arc.

6. Select ***Feature Objects*** | ***Reverse Directions***

7. In the data tree, right-click on the *Drainage* coverage

8. Select *Type…*

9. From the list, select *1D-Hyd Centerline*

The Reverse Directions command changes the direction of the flow of the stream. To view this change, you can zoom in on a small portion of the stream. There are small blue arrows indicating the direction of the flow. The direction should indicate that the stream is flowing down and to the left (southwest).

6. Switch to the *Terrain Data* module

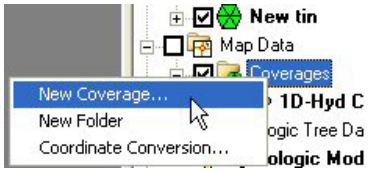
7. Select ***DEM*** | ***Conversion*** | ***DEM -> TIN*** | ***Filtered***

8. Make sure that *Triangulate new TIN* and *Delete DEM* options are toggled on

9. Choose *OK*

10. Select ***Display*** | ***Display Options***

11. On the *TIN* tab, toggle off *Triangles*



12. Select *OK*

13. Right-click on the *Coverages* folder and select *New Coverage…* from the pop-up menu

14. Choose *1D-Hyd Cross Sections* from the *Coverage Type* drop-down box

15. Select *OK*

16. Choose the *Create Feature Arcs* tool

17. Create eight cross-sections as shown in Figure 17-3

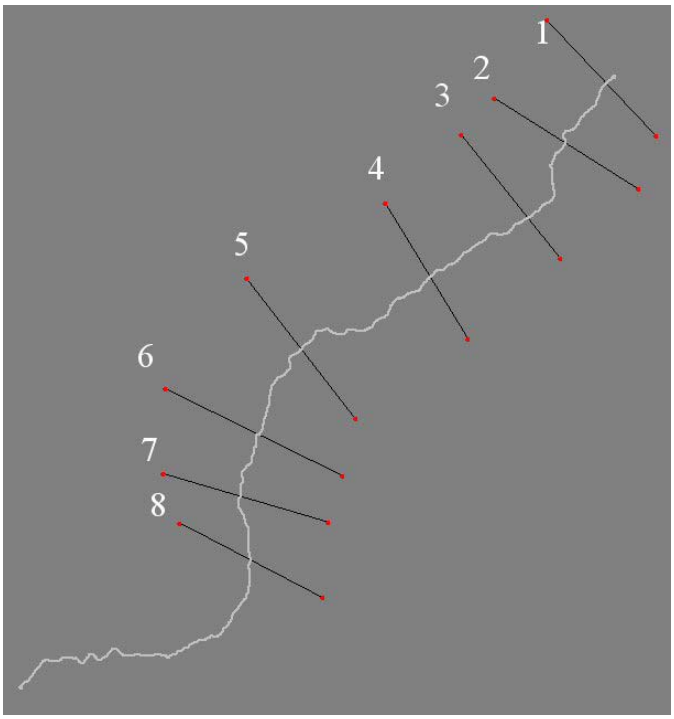


Figure 17-3: Cross Sections on Stream Arc.

17.1.4 Reading in Area Properties

An area property coverage is used to assign Manning's roughness values to the cross sections in SMPDBK. Area property coverages contain polygons with materials (representing land cover types) assigned to each polygon. In this section, you read an existing area property coverage. You could also create your own area property coverage from a background image or map.

1. Select ***File*** | ***Open***
2. Open *areaprop.map*
3. Switch to the *Map* module
4. Choose the *Select Feature Polygon* tool
5. Double-click on the polygons to view the assigned materials

17.1.5 Extracting Cross-Sections

Once you have completed the centerline, cross section, and area property coverages, you are ready to extract the cross sections from the TIN. Then, you must convert your coverage data to a hydraulic model.

1. Click on the *1D-Hyd Cross Section* coverage to make it the active coverage
2. Select ***River Tools | Extract Cross Section…***
3. Toggle on *Using arcs* and select *1D-Hyd Centerline* from the drop-down list
4. Choose *Area Property* from the *Material Zones* drop-down list
5. Select *OK*
6. Save the file as *"xsections"*
7. Choose the *Select Feature Arc* tool
8. Double-click on a cross-section
9. Click on *Assign Xsec* to view the cross-section profile
10. Select *Cancel* twice to exit the dialogs
11. Click on the *1D-Hyd Centerline* coverage to make it the active coverage
12. Select ***River Tools | Map -> 1D Schematic***

17.2 Using SMPDBK

Setting up your hydraulic model geometry is 90% of the work associated with creating a SMPDBK model. The other 10% of setting up your model involves entering information about the dam and entering Manning's roughness values for each of the different area properties. You can find this information on the internet or in the National Inventory of Dams (NID) database. This section will guide you through the process of finishing your SMPDBK model setup.

17.2.1 Edit Parameters

1. Choose the *River* module
2. From the *Model* drop-down box, choose *SMPDBK*
3. Select ***SMPDBK | Edit Parameters …***
4. Enter in values shown in Figure 17-4

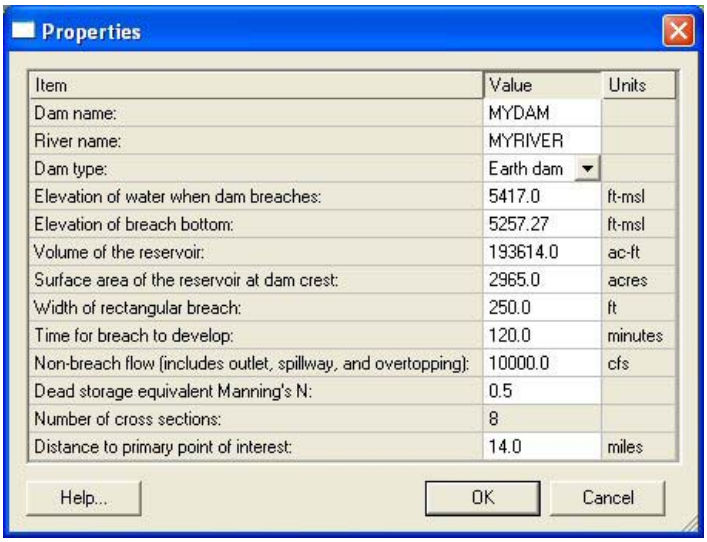


Figure 17-4: Properties Dialog.

5. Select *OK*
6. Select ***SMPDBK | Material Properties …***
7. Enter in the following values:

Forest	0.08
Residential	0.07
River	0.05
Shrub/Brush	0.06

8. Select *OK*
9. Select ***SMPDBK | Model Control …***
10. Choose *Materials* from the drop-down box

11. Select *OK*

12. Select ***SMPDBK | Export SMPDBK File …***

13. Save the file as *smpdbk.dat*. Select *OK* to continue saving your data if any errors are encountered.

17.2.2 Running the Simulation

Your model is now finished and you are ready to run the simulation. When you run the SMPDBK simulation, WMS saves the SMPDBK input file, runs SMPDBK, and attempts to read the SMPDBK solution. A solution point is placed where each cross section intersects the stream centerline in your hydraulic model.

1. Select ***SMPDBK | Run Simulation…***

2. Save the file as *smpdbk.dat*

3. Select *YES* to replace the file. A window will appear and SMPDBK will run in this window.

4. Once SMPDBK finishes running (you may have to wait a few seconds to a minute or so), choose *Close* to close the SMPDBK window. If SMPDBK finishes running successfully, a message such as "Stop—Program terminated" and "SMPDBK Finished" will appear in the model wrapper.

17.3 Post-Processing

Once you have finished running SMPDBK, WMS reads the solution as a 2D scattered dataset. This solution contains water surface elevation points where each cross section intersects your stream centerline. When you delineate the floodplain, you need additional solution points to create a well-defined map. This section will guide you through the processes of interpolating solution points along the centerline and the cross sections. After interpolating to create additional solution points, you will learn how to delineate the floodplain from these points.

17.3.1 Interpolation

1. Switch to the *Map* module

2. Click on the *1D-Hyd Centerline* coverage to make it the active coverage

3. Select ***River Tools | Interpolate Water Surface Elevations …***
4. Select the option to create a data point at a specified spacing (instead of at each arc vertex).
5. Change the *Data point spacing* to 1000
6. Select *OK*
7. Click on the *1D-Hyd Cross Section* coverage to make it the active coverage
8. Select ***River Tools | Interpolate Water Surface Elevations …***
9. Make sure the option is selected to create a data point at a specified spacing
10. Make sure the *Data point spacing* is set to 1000.
11. Select *OK*

17.3.2 Floodplain Delineation

This section will show you how to delineate the flood using the WMS floodplain delineation tools. You will also learn how to adjust the display options to better display the results of the SMPDBK simulation.

1. Switch to the *Terrain Data* module
2. Select ***Flood | Delineate …***
3. Set the *Max search radius* to 5000

4. Select *OK*.
5. Expand the "New tin" Data Tree item to view the datasets associated with the TIN.
6. Expand the *MaxWS (FLOOD)* folder.
7. Select *MaxWS_fd*
8. Select ***File | Open***
9. Open *aerial.jpg*
10. Select ***Display | Display Options …***

11. Click on the *Contours…* button

12. Toggle on *Color fill between contours* and set the transparency to 40%

13. Toggle on the *Display Legend* option

14. Select *OK* twice to exit the dialogs

The flood depths from the SMPDBK simulation can now be viewed as a spatial map. You will notice that some areas appear flooded that you know are not actually flooded if the dam breaches. These areas can be corrected by drawing polygons around the areas you know are not flooded and then re-delineating the floodplain. The following steps explain how to do this.

15. Right-click on the *Coverages* folder in the Data Tree and select *New Coverage…* from the pop-up menu

16. Choose *Flood Barrier* from the *Coverage Type* drop-down box

17. Select *OK*

18. Choose the *Create Feature Arcs* tool .

19. Draw an arc representing a polygon around the extra data that needs to be deleted. This includes areas clearly outside of the floodplain and areas where data does not exist to give accurate results, such as outside the extents of the hydraulic model (see Figure 17-5). WMS will ignore the areas inside this polygon when delineating your floodplain. Be sure your arc forms a closed loop.

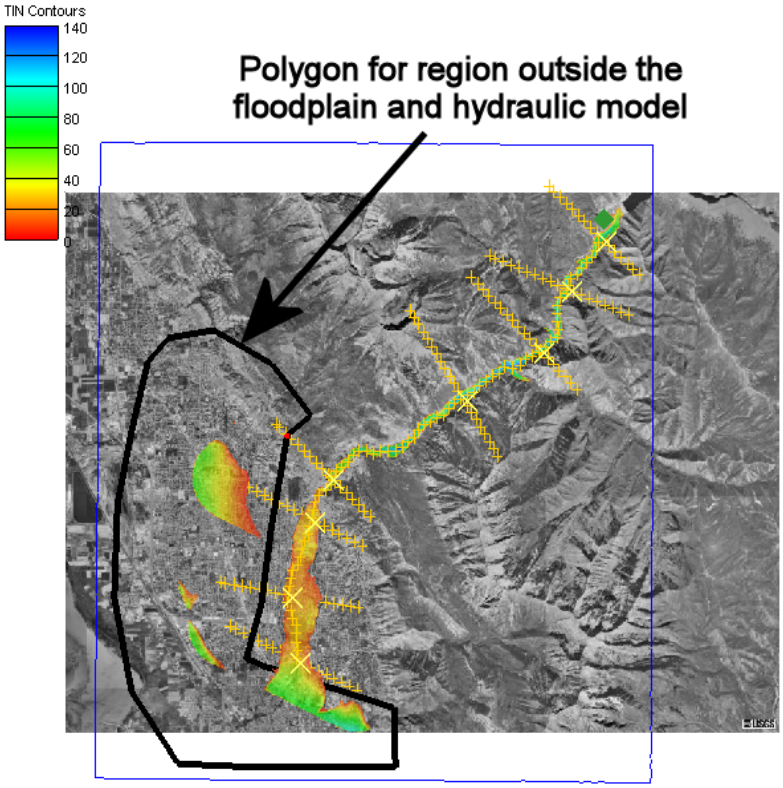


Figure 17-5: Creating a polygon for regions outside the model extents

20. Switch to the *Map* module .
21. Select ***Feature Objects*** | ***Build Polygons***. Select *OK* to use all arcs.
22. Switch to the *Terrain Data* module .
23. Select ***Flood*** | ***Delineate …***
24. Select the *User defined flood barrier coverage* option.
25. Change the solution name to *MaxWS_1*
26. Select *OK*
27. To view the new data, open the MaxWS_1 (FLOOD) solution folder and select MaxWS_1_fd in the Data Tree.
28. You can toggle between MaxWS_fd and MaxWS_1_fd in the Data Tree to view the effects of the flood barrier coverage on the floodplain delineation.

CHAPTER 18

Storm Drain: Rational Design

Storm Drain is a hydraulic analysis and design program for storm drain systems. Developed by the Federal Highway Administration, Storm Drain provides hydraulic engineers with a quick and accurate method of computing the capacity of an existing system, or designing a system to meet a given set of input flows. The Storm Drain interface in WMS uses the same HYDRA program that has been distributed by the FHWA as part of their HYDRAIN computer program package for the past several years.

18.1 Objectives

In this tutorial, we will set up a drainage simulation based on the Rational Method for a proposed subdivision. The objective of this tutorial is to teach you the basic steps for defining a Storm Drain input file, running the numeric model, and viewing the results. These steps include the following:

1. Define runoff coefficients
2. Define the drainage area
3. Compute flows for each basin using the Rational Method
4. Import a pipe network
5. Define the necessary parameters for the network

6. Link nodes from the drainage network to their corresponding nodes in the pipe network
7. Assign elevations to the pipe network
8. Save the Storm Drain input file and run the simulation
9. View the solution

18.2 Defining Runoff Coefficients

18.2.1 Reading in the Subdivision Layout and Elevations

We will open a shapefile containing the geography of the proposed subdivision. Also, to help us in defining elevations and slopes for the drainage area, we will open a TIN (Triangulated Irregular Network) for the area:

1. Select ***File*** | ***Open…***
2. Open "*cougarestates.tin*"
3. Right-click on the *Coverages* folder in the Data Tree
4. Select *New Coverage* from the pop-up menu
5. Change the Coverage type to *Runoff Coefficient*
6. Select *OK*
7. Select ***File*** | ***Open…***
8. Open "*cougarestates.shp*"
9. Select *OK* to import the polygon shapefile

18.2.2 Defining the Runoff Coefficients

We will turn off the display of the TIN in order to better distinguish the subdivision polygons.

1. Switch to the Terrain Data module
2. Expand the *Terrain Data* folder in the Data Tree if necessary
3. Toggle the visibility check box for the *"new" TIN* off in the Data Tree

4. Switch to the Map module
5. Choose the *Select Feature Polygon* tool
6. Double-click on the lower left polygon
7. Enter a runoff coefficient of **0.6**
8. Select *OK*
9. Assign Runoff Coefficients for the remaining polygons as shown in Figure 18-1

Figure 18-1: Assigning runoff coefficients.

18.3 Defining the Drainage Area

First, we will initialize the Storm Drain interface. Initializing Storm Drain tells WMS to make available additional features needed to build a Storm Drain model. Then we will read in pre-defined drainage boundaries and set up drainage inlets and a drainage outfall.

18.3.1 Initializing Storm Drain

1. Change the Model drop-down box located at the top of the WMS interface to *Storm Drain*

2. Select ***Storm Drain*** | ***New Simulation***

18.3.2 Creating the Drainage Coverage

1. Right-click on the *Coverages* folder in the Data Tree
2. Select *New Coverage* from the pop-up menu
3. Change Coverage type to *Drainage*
4. Select *OK*
5. Select ***File*** | ***Open…***
6. Open "*runoff.shp*"
7. Click on the browse button located under the check box for opening a shapefile of arcs
8. Open "*drainage.shp*"
9. Select *OK* to import the shape file data. Figure 18-2 shows what your layout should look like.

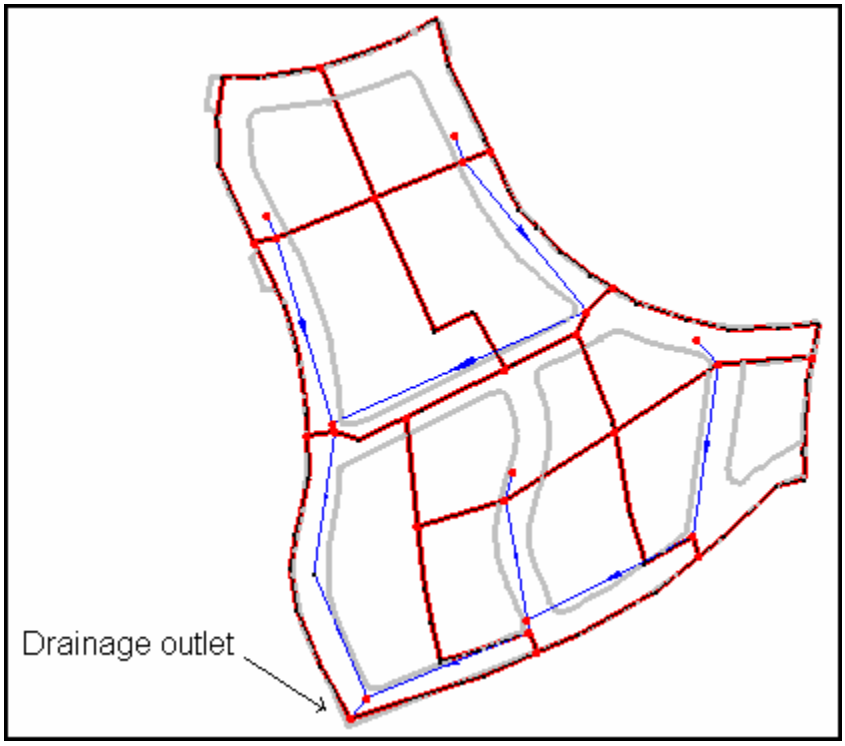


Figure 18-2: Drainage basins and streams for the subdivision.

Before defining nodes as storm drains/drainage outlets, we will hide the Runoff Coefficient coverage in order to simplify the screen display

1. Toggle the visibility check box for the Runoff Coefficient coverage off in the Data Tree

2. Choose the *Select Feature Point/Node* tool

3. Double-click on the node labeled Drainage Outlet in Figure 18-2

4. Change the Point type to *Drainage Outlet* and select *OK*

5. While holding the <SHIFT> key, select each of the nodes labeled as Storm Drains in Figure 18-3. These points correspond to access hole locations in the pipe network. Be sure to select the nodes that intersect with the basin boundary lines.

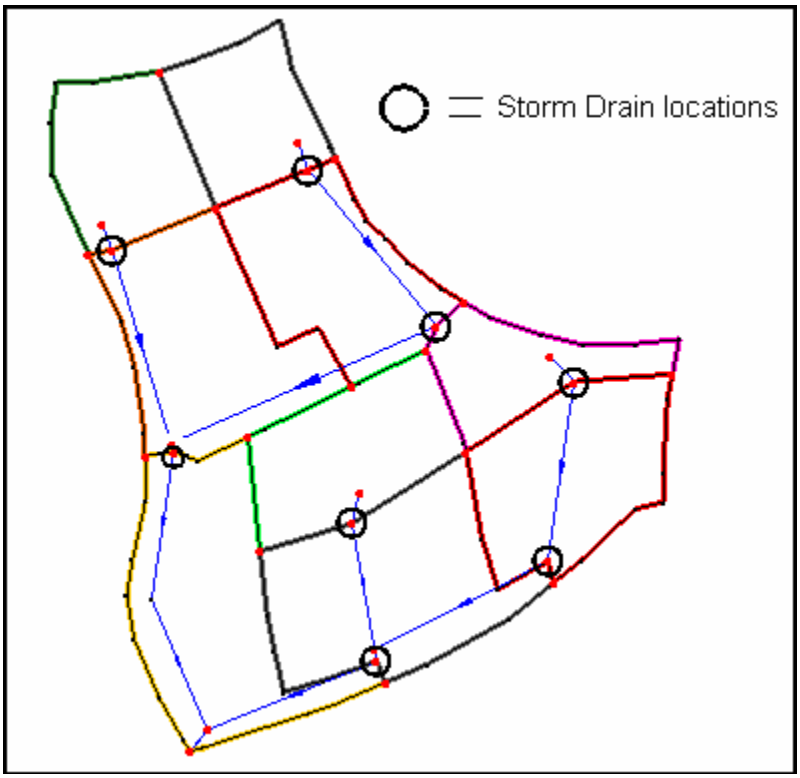


Figure 18-3: Storm Drain node locations.

6. Select ***Feature Objects*** | ***Attributes…***

7. Change the Point type to *Storm Drain Inlet*

8. Click the *Properties* button

9. Change the Inlet type to *Curb opening in sump condition*

10. Enter the dimensions shown in Table 18-1

Table 18-1: Values for the curb parameters.

Length	Width of Depression	Height	Inlet depression
4.0	3.0	1.0	0.25

11. Select *OK* twice to exit both dialogs

12. Select ***Feature Objects*** | ***Compute Basin Data…***

13. Click the *Current Coordinates…* button

14. Set the horizontal and vertical units as *U.S. Survey Feet*

15. Select *OK*

16. Set Basin Areas to *Acres* and Distances to *Feet*

17. Select *OK*

18.3.3 Computing Runoff Coefficients

Composite runoff coefficients must be computed for each drainage area. A weighted average for each drainage area is calculated from data in the Runoff Coefficient coverage. To compute the basin runoff coefficients:

1. Switch to the Hydrologic Modeling module

2. Select ***Calculators*** | ***Compute GIS Attributes…***

3. Set the Computation type as *Runoff Coefficients*

4. Select *OK*

Composite runoff coefficients for each drainage area are computed and displayed on the screen.

18.3.4 Entering Times of Concentration

The Time of Concentration for a basin can be calculated using a Time Computation Coverage and is discussed in more detail in another WMS tutorial. For this tutorial, however, it is assumed that the TC values have already been calculated, and we will input them manually.

1. Change the Model drop-down box (located towards the top of the WMS interface) to *Rational*

2. Choose the *Select Basin tool*

3. Double-click on the basin icon for the basin in the lower left-hand corner of the subdivision

4. Enter **14** for the Time of Concentration (minutes)

5. Select each of the remaining drainage basins and assign Time of Concentration values using Figure 18-4 as a guide

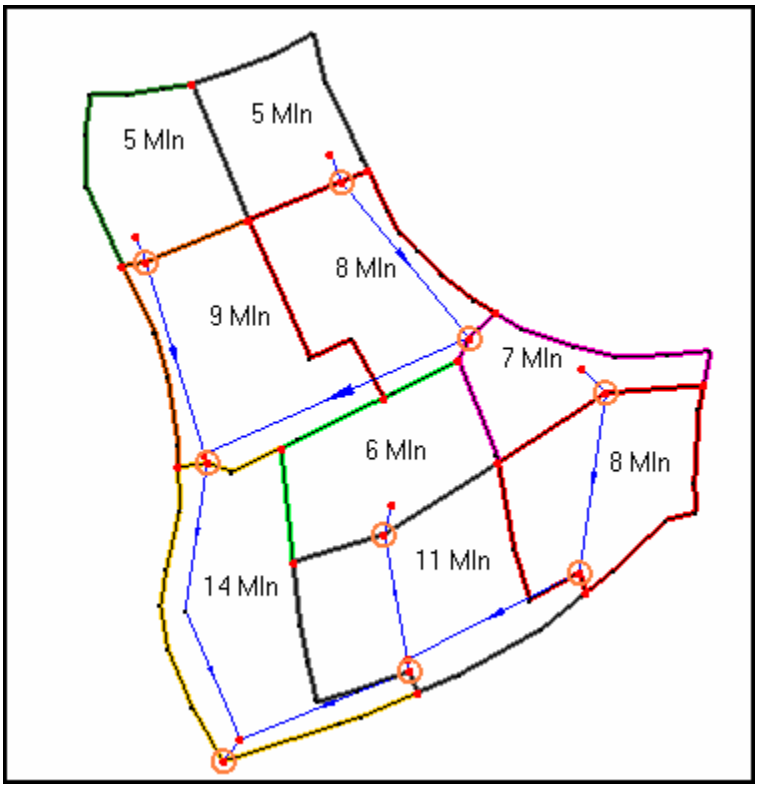


Figure 18-4: TC values for the drainage areas.

6. Select *Done* to close the Rational Method dialog

We have now set up a traditional rational simulation for the Cougar Estates subdivision. The only remaining step is to define the IDF curves. This step will be included as part of the Storm Drain process. We will now import a pre-defined pipe network, assign properties to the network, and link it to the Drainage coverage.

18.4 Importing the Pipe Network

As mentioned above, we will be importing a pipe network to use in our Storm Drain simulation. This network was defined as a shapefile, and will be converted to feature objects by WMS. WMS can also open DXF files and

convert them to feature objects. Alternatively, users can manually create a pipe network using the *Create Feature Arc* tool .

1. Switch to the *Map* module
2. Right-click on the *Coverages* folder in the Data Tree
3. Select *New Coverage*
4. Change the Coverage type to *Storm Drain*
5. Select *OK*
6. Select ***File*** | ***Open…***
7. Open "*pipenetwork.shp*"
8. Select *OK* to import the shapefile

18.4.1 Entering Job Control Data

1. Select ***Storm Drain*** | ***Job Control***
2. Enter **Storm Drain Rational** for the Title
3. Choose the *English* option for the Units
4. Click on the *Pipe Data Properties* button

This dialog allows users to define a set of pipe properties that will be applied to all pipes in the network. In addition to assigning pipe properties on a global scale, users can define specific properties to individual pipes, or to a group of pipes by selecting them with the *Select Feature Arc* tool and selecting the Attributes command from the Feature Objects menu. For this tutorial, we will simply define global pipe properties.

5. Change the Pipe type to *Concrete-Smooth*

When a user specifies a pipe type, WMS fills in an appropriate Manning's n roughness value based on that pipe type. Users can also enter/edit the Manning's n value directly in this dialog. Users can change the Manning's n values that are assigned to the various Pipe Types. These values can be accessed and edited by opening the Display Options dialog, choosing the Map tab, and changing the Active Coverage field to *Storm Drain*.

6. Enter the values displayed in Figure 18-5

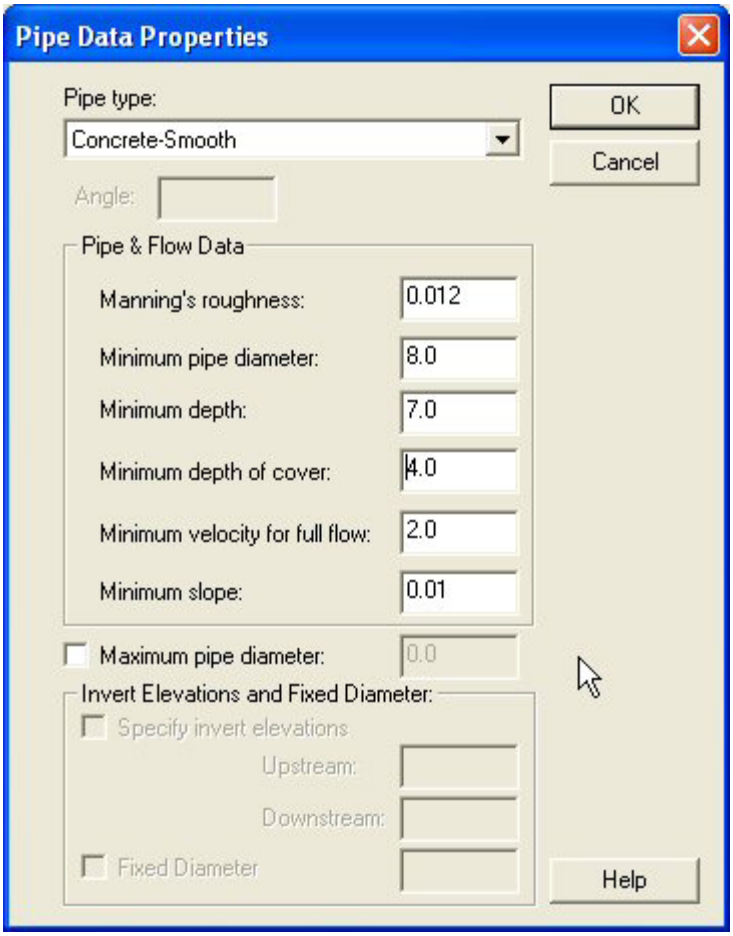


Figure 18-5: Pipe property values for the pipe network.

7. Select *OK* to exit the Pipe Data Properties dialog

18.4.2 Defining Rainfall Data

1. Click on the *Rainfall Data* button
2. Select the *Run Compute IDF Curve Dialog* option
3. Select *OK*
4. Choose the *User Supplied Data* option as the IDF Curve Computation
5. Click the *Define Data* button
6. Change the Recurrence value to *25*
7. Enter the precipitation values shown in Figure 18-6

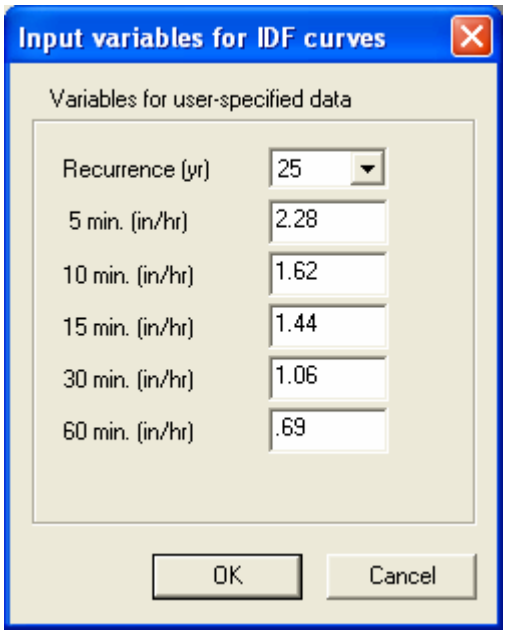


Figure 18-6: Values for computing the 25 yr IDF curve.

8. Select *OK*

9. Highlight the line corresponding to the *25-yr* precipitation values from the window in the upper right-hand corner of the dialog

10. Select *Done*

11. Select *OK* twice more to return to the WMS interface

12. Switch to the Hydrologic Modeling module

13. Choose the *Select Basin* tool

14. Double-click on the basin labeled 4B

Notice that the intensity value computed in the Storm Drain interface has been copied here as well, and Q is now computed for the basin.

15. Select *Done* to exit the Rational Method dialog

18.4.3 Defining the Outfall and Access Hole Locations

1. Switch to the Map module

2. Make sure that the *Storm Drain* coverage is selected in the Data Tree

3. Choose the *Select Feature Point/Node* tool

4. Double-click on the node in the lower left-hand corner of the subdivision as shown in Figure 18-7

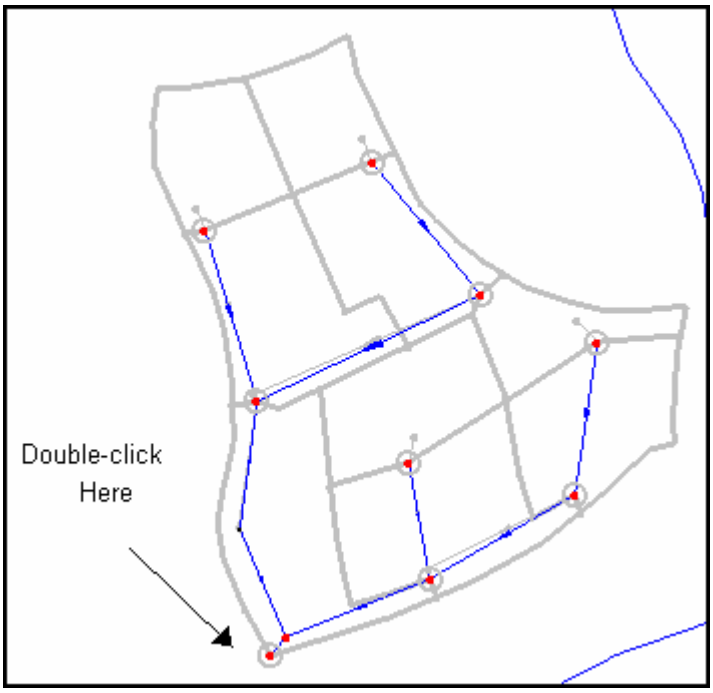


Figure 18-7: Outfall location.

5. Change the node type to *Outfall*

6. Select *OK*

7. Multi-select the nodes indicated in Figure 18-8 by holding down the <SHIFT> key as you select each node

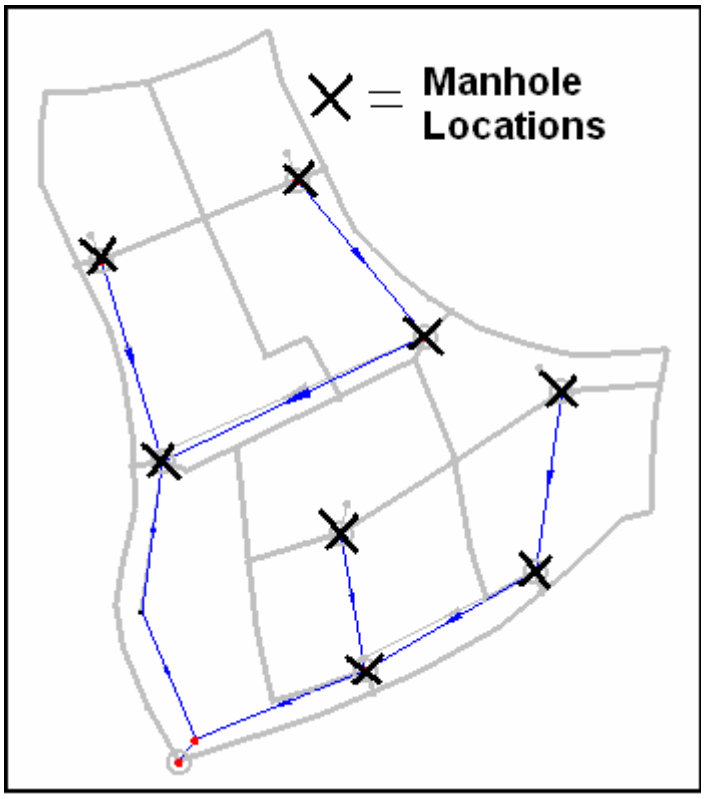


Figure 18-8: Access hole locations.

8. Select ***Feature Objects*** | ***Attributes…***
9. Change the node type to *Access Hole*
10. Enter **4.0** as the Acess hole width
11. Change the Bench Type to *Flat Bench*
12. Select *OK*

The Pipe Drain Network is now defined

18.5 Linking Nodes and Assigning Elevations

To properly portray the relationship between the Drainage coverage and the Storm Drain coverage, we need to explicitly define links between their corresponding nodes. We also need to assign elevations to all linked nodes. Elevations can either be defined manually, or with the use of a background TIN or DEM. We will use the TIN that we opened earlier in this tutorial for assigning elevations.

1. Select ***Storm Drain*** | ***Link Nodes…***

2. Click the *Auto Link* button. This will link the storm drain inlets from the Drainage coverage to the access holes in the Storm Drain coverage. All nodes should be linked except the one node just upstream of the outfall. This node is a pipe junction and is not associated with a storm drain inlet.

3. Select *OK*

4. Select ***Storm Drain*** | ***Assign Pipe Elevations***. This command automatically assigns elevations to nodes based on an underlying TIN or DEM, if one is present.

With Drainage and Storm Drain nodes linked together and elevations assigned to our pipe network, we are ready to save the Storm Drain model.

18.6 Saving the Simulation and Running Storm Drain

1. Select ***File*** | ***Save As…***

2. Make sure the Save as type filter is set to *WMS Project Files (*.wpr)*

3. Enter **wms_storm_rat** for the File name

4. Select *Save*

5. Select ***Storm Drain*** | ***Save Simulation***

6. Save the file as **wms_st_r**

7. Select ***Storm Drain*** | ***Run Simulation***

8. Verify that the input file name is "*wms_st_r.hda*"

9. Change the output file name to "*wms_st_r.lst*"

The input and output file name prefixes can be up to 8 characters in length. If you enter an input file name longer than 8 characters, Storm Drain will not run. If you enter an output file name longer than 8 characters, it is truncated to 8 characters when Storm Drain writes the file. Also all directory names in the path of the input/output files cannot contain more than 8 charcters.

10. Select *OK*

11. Once Storm Drain (HYDRA) has finished running, select *Close*

12. Select ***File*** | ***Edit File…***

13. Open "*wms_st_r.lst*"

14. Select *OK* to open the file with **Notepad**

By browsing through this file, you will see all of the parameters calculated by the Storm Drain (HYDRA) model, such as recommended pipe diameters, pipe invert elevations, flows, velocities, and hydraulic grade line computations.

15. Close Notepad

CHAPTER 19

Storm Drain: Hydrographic Design

Storm Drain is a hydraulic analysis and design program for storm drain and sanitary sewer systems. It was developed by the Federal Highway Administration with the intent of providing hydraulic engineers a quick and accurate method of designing and analyzing storm drain, sanitary sewer, and combination systems. The Storm Drain interface in WMS uses the same HYDRA program to perform calculations that is currently part of the FHWA's HYDRAIN suite of hydraulic programs.

This tutorial demonstrates how to use WMS to set up a Hydrographic Drainage simulation for use with Storm Drain. We will set up drainage areas and Runoff Coefficients for a subdivision, similar to the one in the previous tutorial, but this time we will also specify gutter locations in the Drainage coverage. As illustrated in this tutorial, the pipe network does not need to flow parallel or in the same direction as the over-lying drainage. However, there are some limitations in the way the drainage coverage is constructed in order to run a hydrographic analysis with Storm Drain. The most important thing is that each inflow hydrograph needs to be associated with one and only one gutter at the storm drain inlet. In other words "extra" stream arcs that are not part of the gutter leading into the inlet should be deleted.

In this analysis we will generate input hydrographs using the Rational Method calculator, but it should be emphasized that any hydrologic method that produces a hydrograph can be used, or hydrographs may be entered manually.

19.1 Objectives

In this tutorial, we will set up a hydrographic storm drain simulation for a hypothetical subdivision. The objective of this tutorial is to teach you the basic steps for defining a Storm Drain input file for hydrographic analysis. These steps include the following:

1. Build the drainage basins and make adjustments for limitations in doing a hydrographic analysis
2. Perform a simple Rational Method analysis to generate input hydrographs
3. Digitize a small pipe network based on the surface drainage
4. Define the necessary parameters for the pipe network
5. Run Storm Drain (HYDRA)
6. View the solution

19.2 Developing the Surface Drainage Coverage

This tutorial will begin with a drainage coverage created with techniques typical to those described in earlier tutorials. In order to prepare the drainage coverage for storm drain analysis we will eliminate all of the stream arcs, except ones that lead into a given outlet (a storm drain inlet in this case). Having extra branching streams does not allow WMS to build the storm drain input file correctly. So you must decide which arc represents the primary stream (in this case a gutter) into an inlet and then delete the others.

1. Switch to the Map module
2. Select ***File*** | ***Open…***
3. Open "*sdstart.map*"

19.2.1 Eliminating Unnecessary Stream Arcs and Nodes

Figure 19-1 shows the drainage coverage, with the extra streams highlighted. In order to prepare the drainage coverage correctly you will need to delete these stream arcs.

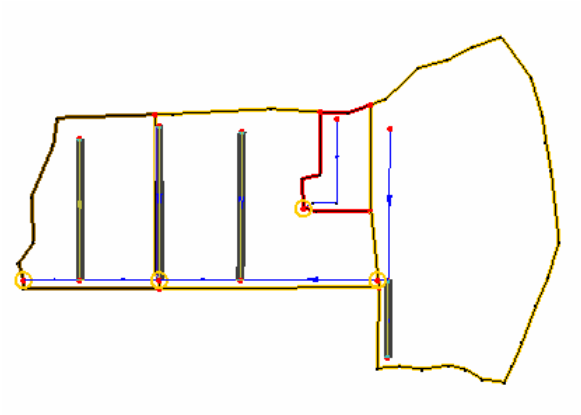


Figure 19-1: Initial drainage coverage with unnecessary stream arcs highlighted.

1. Select the *Select Arc* tool

2. While holding down the <SHIFT> key (for multi-selection) select the four arcs highlighted in Figure 19-1

3. Select the *Delete* Macro (or Edit | Delete)

4. Select *OK*

You must now delete the extra nodes that were at the intersection of the arcs just deleted and the remaining arcs so that there is just a single arc entering the storm drain inlet points.

5. Select the *Select Feature Point/Node* tool

6. While holding down the <SHIFT> key select the three nodes identified in Figure 19-2 (be sure to select the point to the right of the outlet at the far right and not the point that is the outlet)

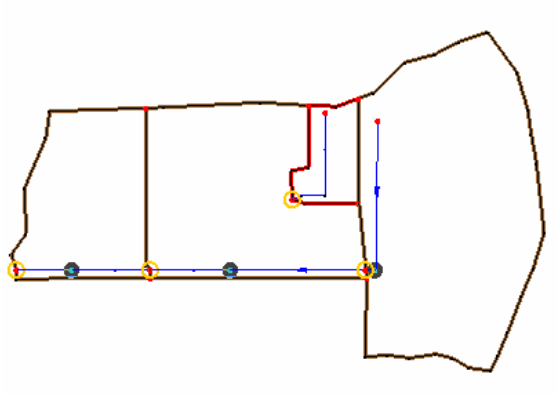


Figure 19-2: Nodes that need to be converted to vertices.

7. Select ***Feature Objects | Vertex <-> Node***

19.2.2 Initializing Storm Drain Simulation

In order to assign the remaining stream arcs to be gutters and the outlet points as storm drain inlets we must initialize a storm drain analysis.

1. Select *Storm Drain* from the drop down list of models in the Edit window

2. Select ***Storm Drain | New Simulation***

19.2.3 Assigning Stream Arcs as Gutters

1. Select the *Select Arc* tool

2. While holding down the <SHIFT> key, select all four stream arcs as indicated in Figure 19-3

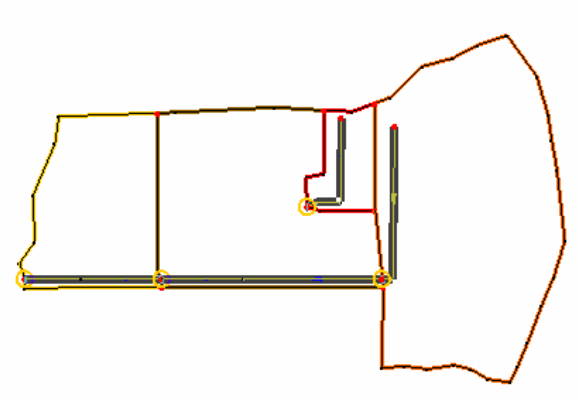


Figure 19-3: Arcs to be selected.

3. Select ***Feature Objects | Attributes…***

4. Select *Gutter* for the Feature Arc Type

5. Choose the *Atts* button to define the properties of the gutters

6. Set the type to *Uniform Gutter* (should be the default) and set the Manning's N to **0.015**, the Roadway Cross-slope to **0.02**, and the width to **2.0**

7. Select *OK*

8. Select *OK*

19.2.4 Assigning Outlets as Storm Drain Inlets

For this analysis two of the outlets will be defined in areas of depression (sump conditions) while the other two will have bypass flow (on grade conditions).

1. Select the *Select Feature Point/Node* tool

2. While holding down the <SHIFT> key select the outlet point of the entire basin system, and the outlet of the isolated interior basin as indicated in Figure 19-4

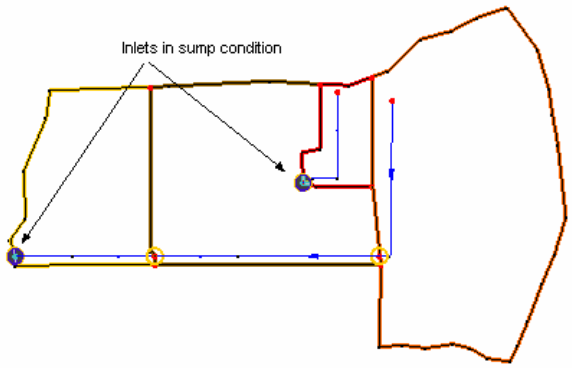


Figure 19-4: Outlet points to be selected.

3. Select ***Feature Objects | Attributes…***

4. Set the Drainage Feature Point Type to be *Storm Drain Inlet*

5. Choose the *Properties* button

6. Set the Type to be *Grate in Sump Condition*, the Subtype to be *Reticuline*, the width to be **2.0**, the length to be **6.0**, the inlet perimeter to be **16.0**, and the area to be **8.5**

7. Select *OK*

8. Select *OK*

9. Select one of the other two remaining outlets (this will insure that the two selected outlets become unselected)

10. Now while holding down the <SHIFT> key select the remaining outlet (the one that has not yet been selected)

11. Select ***Feature Objects | Attributes…***

12. Set the Drainage Feature Point Type to be *Storm Drain Inlet*

13. Choose the *Properties* button

14. Set the Type to be *Grate on grade*, the Subtype to be *Reticuline*, the width to be **2.0**, and the length to be **6.0**

15. Select *OK*

16. Select *OK*

17. Select ***Feature Objects | Compute Basin Data…***

18. Select *OK*

19.3 Running a Rational Analysis

We will perform a simple Rational analysis of this subdivision and use the Rational method hydrographs as input for the hydrographic storm drain analysis. You should remember though that any of the hydrologic analysis models in WMS can be used, or input hydrographs may be entered manually.

1. Select the Hydrologic Modeling module

2. Change the Model drop-down box located at the top of the WMS interface to *Rational*

19.3.1 Defining Rational Method Basin Parameters

1. Double-click on one of the basin icons using the *Select Basin* tool and define the basin parameters according to Figure 19-5 and Table 19-1. Select another basin and define its parameters until all basin data is defined. For more details on defining a Rational Simulation see the tutorial on Rational method analysis.

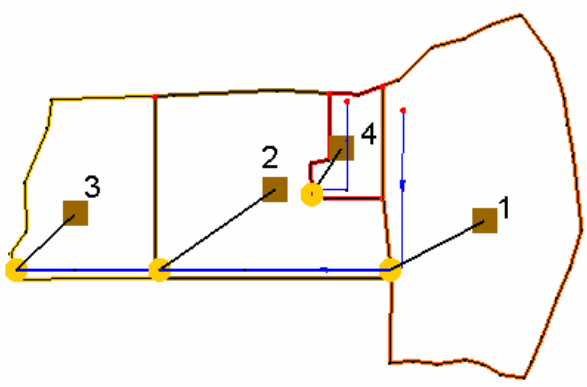


Figure 19-5: Basin locations.

Table 19-1: Rational Method basin parameters.

Basin	C	I (in/hr)	Tc (min)
1	.1	2.4	22
2	.15	2.7	18
3	.15	3.1	12
4	.2	3.8	7

2. Select *Done* in the Rational Method dialog

19.3.2 Defining Rational Method Routing Parameters

1. Define the intensity and routing time (where not dimmed) for the four confluence points connected to the basins according to Table 19-2. As with basins, reach data are entered by double-clicking on the yellow outlet icons using the *Select Outlet* tool .

Table 19-2: Rational Method routing parameters.

Outlet	Routing Lag (min)	I (in/hr)
1	5	2.4
2	4	1.9
3	--	1.7
4	--	3.8

2. Select *Done* in the Rational Method dialog

19.3.3 Computing Hydrographs

1. Double-click on the outlet for the isolated interior basin (the outlet of basin 4 in Figure 19-5

2. Choose the *Define Hydrographs* button

3. Select *Done* (use the default settings for the Rational hydrograph method)

4. Select *Done*

5. Double-click on the outlet for the entire watershed (the outlet of basin 3 in Figure 19-5)

6. Choose the *Define Hydrographs* button

7. Select *Done* (use the default settings for the Rational hydrograph method)

8. Select *Done*

At this point you should have hydrographs defined for the drainage coverage. These hydrographs will be used as input to the storm drain problem.

19.4 Creating the Pipe Network

You will now create a storm drain pipe network that connects the isolated interior basin to the others and exits west of the watershed outlet. You will do this by creating a Storm Drain coverage and then digitizing the storm drain pipes using the *Create Feature Arc* tool .

19.4.1 Digitizing Storm Drain Pipes

1. Switch to the Map module

2. Expand the *Map Data* folder in the Data Tree (if it isn't already expanded)

3. Right-click on the *Coverages* folder in the Data Tree window

4. Select *New Coverage*

5. Change the Coverage type to *Storm Drain*

6. Select *OK*

7. Select the *Create Feature Arc* tool

8. Beginning at a point just to the left (west) of the outlet of the watershed (basin 3) begin a pipe network segment and end it by double-clicking on top of the outlet of basin 3.

9. Now complete the storm drain network by creating the pipe segments from downstream to upstream between basin 3 and basin 2, basin 2 and basin 1, and a connection from the pipe between basin 2 and basin 1 to the outlet of basin 4. Be sure that feature nodes (segment endings) are created by double-clicking to end the arc to correspond with the storm drain inlets of the drainage coverage. Your storm drain coverage should look like the one in Figure 19-6.

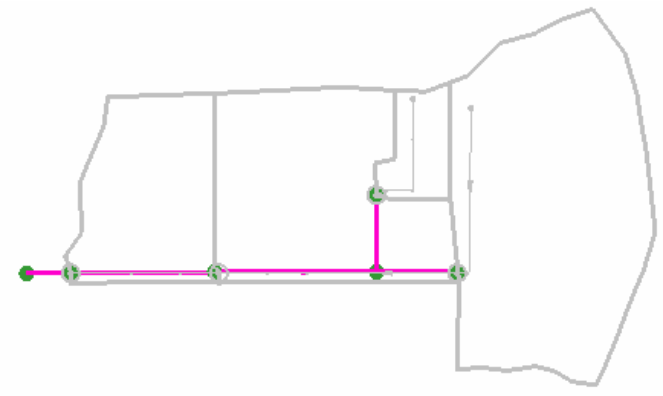


Figure 19-6: View of the Storm Drain coverage.

19.4.2 Entering Storm Drain Parameters

1. Select ***Storm Drain*** | ***Job Control***
2. Set the title to: **Hydrographic Storm Drain Analysis**.
3. Set the Criteria Switch to *3 - Storm (Hydrographic) only*
4. Set the timestep to **2**
5. Set the Units to *English*
6. Choose the *Pipe Data Properties* button
7. Set the values according to Figure 19-7

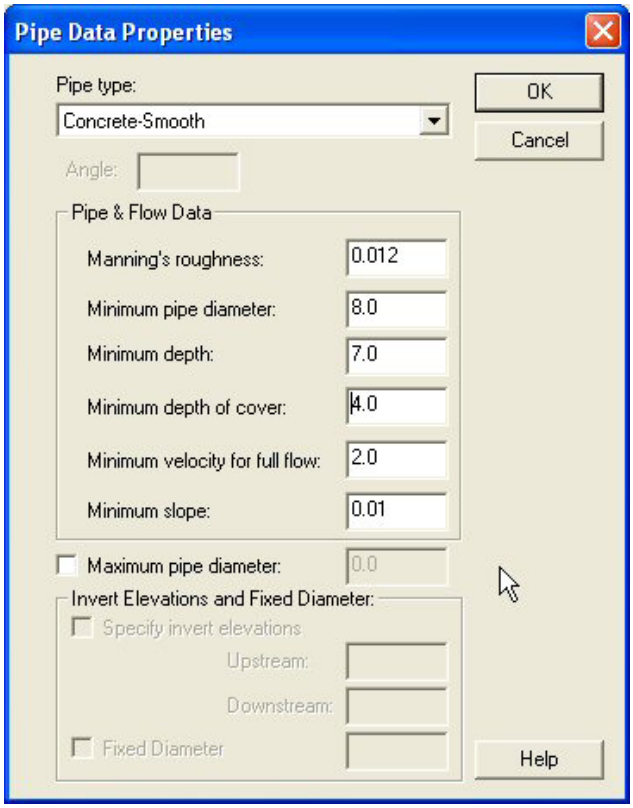
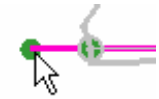


Figure 19-7: Pipe properties.

8. Select *OK*

9. Select *OK*

19.4.3 Defining the Outfall and Access Hole Locations



1. Choose the *Select Feature Point/Node* tool

2. Double-click on the node to the left of the drainage basin

3. Set the Node type to be *Outfall*

4. Select *OK*

5. While holding down the <SHIFT> key select the four nodes of the storm drain coverage that correspond to the 4 storm drain inlets (access holes) of the drainage coverage as shown in Figure 19-8

6. Select ***Feature Objects | Attributes…***

7. Set the Node type to be *Access Hole*

8. Set the width to be **4.0**

9. Select *OK*

Of course you could define each access hole separately by double-clicking on the node and setting the node type and access hole width, but it is more efficient to define them all at once. At this point you should have one outfall node, four access hole nodes, and a single pipe junction in your storm drain coverage as identified in Figure 19-8. If this is not the way your storm drain coverage is set up, then correct it now.

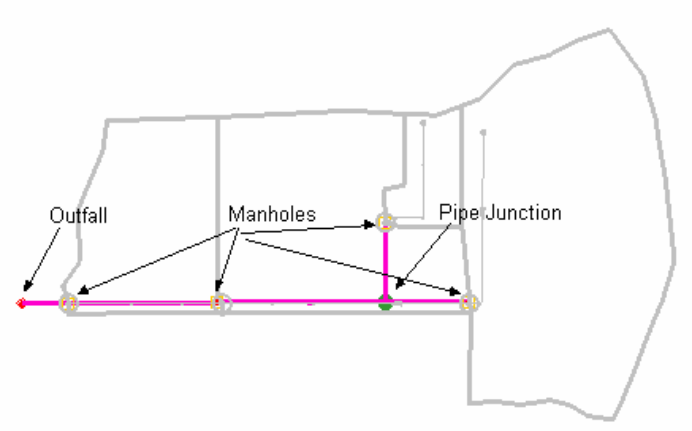


Figure 19-8: The correct setup of the Storm Drain coverage.

19.4.4 Linking Nodes

To properly portray the relationship between the drainage coverage and the storm drain coverage, we need to explicitly define links between their corresponding nodes.

1. Select ***Storm Drain*** | ***Link Nodes…***

2. Change the tolerance to **25**

3. Click the *Auto Link* button. This should link the four storm drain inlet nodes of the drainage coverage to the four access holes of the storm drain coverage. If not, increase the tolerance by an increment of **5** and select the *Auto Link* button until the four node pairs are linked.

4. Select *OK*

19.4.5 Assigning Elevations and Mapping Hydrographs

We also need to assign elevations to all the nodes in the Storm Drain and Drainage coverages. Elevations can either be defined manually, or with the use of a background TIN or DEM. In this exercise we will manually define them.

1. Toggle the display of the Drainage coverage off in the Data Tree

2. Choose the *Select Feature Point/Node* tool

3. Set the elevations of the nodes in the Storm Drain coverage according to the values given in Figure 19-9. To set an elevation select the node and change the elevation in the *Z:* edit field in the Edit window at the top of the WMS interface.

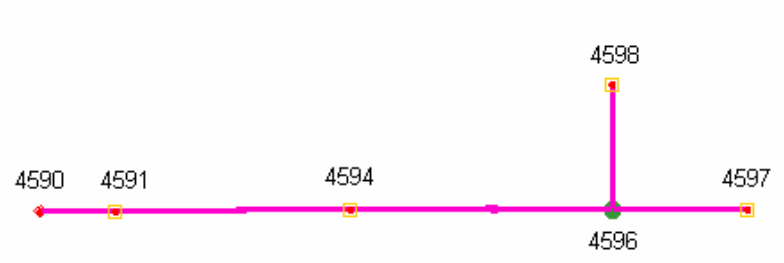


Figure 19-9: Node elevations for the storm drain pipes.

4. Toggle the *Drainage* coverage on and the *Storm Drain* coverage off in the Data Tree

5. Select the *Drainage* coverage in the Data Tree so that it is the active coverage

6. Set the elevations of all the nodes connected to the gutter arcs according to Figure 19-10

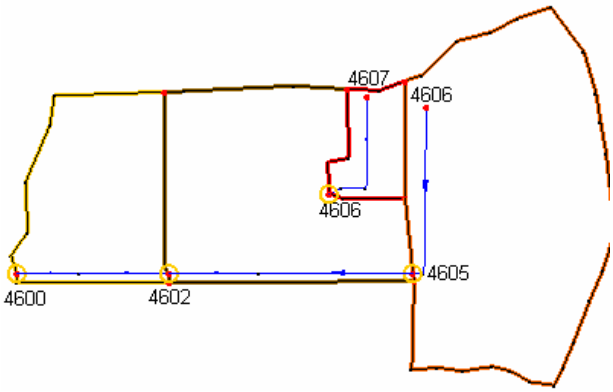


Figure 19-10: Node elevations for the gutters.

7. Toggle the Storm Drain coverage on in the Data Tree and select it so that it is the active coverage

8. Select ***Storm Drain | Map Hydrographs***

With Drainage and Storm Drain nodes linked together and elevations assigned to our pipes and gutters, we are ready to save and run the Storm Drain model.

19.5 Saving the Simulation and Running Storm Drain

1. Select ***File*** | ***Save As…***
2. Make sure the *Save as type* filter is set to *WMS Project Files (*.wpr)*
3. Enter **wms_storm_hyd** as the File name
4. Select *Save*
5. Select ***Storm Drain*** | ***Save Simulation***
6. Save the file as "*wms_st_h"*
7. Select ***Storm Drain*** | ***Run Simulation***
8. Set the input file name to "*wms_st_h.hda*" and the output file name to "*wms_sd_h.lst*."

The input and output file name prefixes can be up to 8 characters in length. If you enter a name longer than 8 characters, it is truncated to 8 characters when Storm Drain writes the file. Further no directory in the path to the files can be longer than 8 characters.

9. Select *OK*
10. Once Storm Drain (HYDRA) has finished running, select *Close*
11. Select ***File*** | ***Edit File…***
12. Open "*wms_sd_h.lst*"
13. Select *OK* to open the file with **Notepad**

By browsing through this file, you will see all of the parameters calculated by the Storm Drain model, such as recommended pipe diameters, pipe invert elevations, flows, velocities, and hydraulic grade line computations.

14. Close Notepad

CHAPTER 20

HSPF Interface

This chapter demonstrates how WMS can be used to process digital elevation and land use data to develop an HSPF input (.uci) file. The graphical user interface used to define input parameters is demonstrated for a basic hydrology simulation that includes doing the following:

- Delineating watershed segment boundaries from a digital terrain model and USGS land use file.
- Defining segment parameters for a hydrologic analysis.
- Developing reach segment parameters.
- Defining precipitation time series data from standard .wdm database files.
- Entering mass links to define transformations from basin to reach.

20.1 Opening the Watershed and Initializing the HSPF Model

You will open a watershed that has been delineated from a USGS 7.5' DEM and converted to a TIN.

1. Select ***File*** | ***Open…***
2. Locate and choose the file named: *littlecotton.wpr*. This file should be in the \tutorial\CH18\ directory.

3. Select *OK*

The watershed will appear on your screen with the area displayed in the center of the basin. This indicates that the Basin Data has been computed and you are ready to set up the HSPF model. To initialize the model:

1. Switch to the *Hydrologic Tree* module
2. Choose *HSPF* from the *Model* drop down list at the Menu bar.
3. Select ***HSPF*** | ***New Simulation***
4. Select ***HSPF*** | ***Global Options***

You will now enter the parameters to indicate that the model will simulate about 3 years of time with a time step of 1 day. You will also name the model, choose the units to use, specify output levels, and name the input/output files:

1. Enter Little Cottonwood Canyon Model in the Title field.
2. Click on the *Start Time* button; enter *Year*: 1996, *Month*: 1, *Day*: 1, *Hour*: 0, *Minute*: 0, *Second*: 0.0.
3. Click *OK*; the start date will appear to the right of the *Start Time* button.
4. Click on the *End Time* button; enter *Year*: 1999, *Month*: 7, *Day*: 31, *Hour*: 0, *Minute*: 0, *Second*: 0.0.
5. Click *OK*; the end date will appear to the right of the *End Time* button.
6. Enter 24 in the *hours* field under the *Time step* heading (leave minutes at 0)
7. Ensure that the *Units flag* is set to English
8. Change the *Run Flag* to 1; this indicates that HSPF will interpret the data and run the simulation (0 indicates that HSPF will only interpret – error check- the data).
9. Set the *OUTLEV* to 10 (under *Output Levels*); this indicates maximum output to the Error and Warning files. Leave the *SPOUT* flag at 0 (this is only for Special Actions).
10. In the *Files* section, enter *littleco* in the *Prefix* field, then click the *Update Filenames* button. This names all the HSPF input/output files associated with this model to the same name.

Note: you have just indicated that you will be using a file named "*littleco.wdm*" for time series input and output – this file must be created and named appropriately outside of WMS (this has been done for you in this case).

11. Select *OK*

The Global Options are now set for your model. You are now ready to proceed to segmenting the watershed and entering parameters for the segments.

20.2 Importing Land Use and Segmenting the Watershed

To divide this watershed into hydrologically similar segments, you will overlay a land-use data layer. You will read this data from an ArcView Shapefile:

1. Select ***File*** | ***Open…*** Make sure the \tutorial\CH18\ directory is selected.

2. Select the file named *littleclanduse.shp* and click *Open*

3. When the *Import Shapefile Data* dialog appears, click on the *Active Coverage Options* button

4. In the coverage properties dialog, change the coverage type to *Land Use*

5. Click *OK* to return to the *Import Shapefile Data* dialog; note that the land use coverage is listed by the *Active Coverage Options* button.

6. Click on the *Attribute Mapping* button under *Polygons*. This will allow you to choose the database fields that correspond to WMS attributes.

7. Choose *LUCODE* in the *Database fields* window and *Land Use* in the *Coverage attributes* window, then click on the *Map* button. You will see a new line appear in the lower window.

Note: You may get a warning message if LUCODE and Land Use are already mapped. If you do get that warning, click *OK* and you will see *LUCODE=> Land Use* line in the lower window.

8. Choose *PERIMETER=> Basin perimeter* line in the lower window and click *Unmap*. This attribute is recognized automatically by WMS, but is not needed in this case.

9. Choose *Done*

10. Choose *OK* in the *Import Shapefile Data* dialog to read the file

You will see the land use data polygons overlaid on the watershed. You are now ready to compute HSPF segments based on the land use polygons that fall inside the watershed.

1. Select ***Calculators*** | ***Compute GIS Attributes***
2. Choose *HSPF Segments* from the *Computation* drop-down list
3. In the central section of the dialog, ensure that *Land use coverage* appears next to "*Use a"* text and that *LC Land Use* is the selected *Land use coverage name*.
4. A text file that correlates the land use id to land use attributes (name, perviousness, etc.) must be read in. Click the *Import* button to browse for this file. Make sure the \tutorial\CH18\ directory is selected.
5. Choose the file named *littleclanduse.tbl*. Click *Open* to read it into WMS.
6. Choose *OK* in the *Compute GIS Attributes* dialog to compute the HSPF segments
7. Click *OK* when warned that Continuing will delete all HSPF segment data. (If you get that warning)

You should now see the segments computed for the watershed displayed. Each colored area represents a segment to be modeled by HSPF. To better view the segments, you will need to change some display options.

1. Switch to the *Map* module
2. Click on the plus sign "+" next to *Map Coverages* folder in the *Data Tree*. (If it is minus sign "-", proceed to the next step)
3. Click the toggle box next to *LC Land Use* to make it not visible
4. Select ***Display*** | ***Display Options…***
5. Choose the tab entitled *General*
6. Toggle ON the *Land use legend*
7. Click *OK*

You should now see the land use legend which shows the color and title of each segment in the watershed. You may want to Pan and/or Zoom to avoid having the legend appear over the watershed.

20.3 Aggregating Segments

You will note from the display that there are several land segments in the watershed. Some are quite large, such as the Evergreen Forest Land segment, while others are rather insignificant, like Bare Exposed Rock. To make this simulation simpler, you will aggregate some of these segments and delete some others.

1. Switch to the *Tree* module
2. Select the basin icon by clicking on it
3. Select ***HSPF*** | ***Edit Parameters***
4. Note in the *Edit Parameters* dialog the list of segments in the *Basin Data* window. There are 8 separate segments (or land use classifications) in this watershed.

There are two ways to reduce the number of segments computed by WMS. The first is to delete one segment, then add the area of the deleted segment, manually, to another similar segment (aggregate segments). The second is to just delete the segment, then let WMS redistribute the deleted area to the remaining segments (Linear distribution).

1. Choose segment entitled *Other Urban Or Build Up* from the *Basin Data* window. Note that the area is about 35 acres – insignificant in this watershed.
2. Click the *Delete* button
3. Click *Yes* for the message that asks if the segment is to be removed from further use
4. Repeat steps 1, 2 and 3 for the segment entitled *Bare Ground*
5. Click *Done*
6. Switch to the *Drainage* module
7. Select ***TIN*** | ***Compute Basin Data.*** Click *OK* on the *Compute Basin Data* dialog to proceed. This will re-compute basin data and distribute the deleted segments' areas to the remaining segments.
8. Now you will aggregate some segments. Switch back to the *Tree* module .
9. Select the basin icon by clicking on it

10. Select ***HSPF*** | ***Edit Parameters.*** Note that there are only 6 segments listed now.

11. Choose the segment entitled *Mixed Forest*. Note the area of 474.94 acres; this is the area that you will add to the *Evergreen Forest* segment to aggregate the two segments.

12. While highlighting *Mixed Forest* segment, change its area to 0.0 (from 474.94)

13. Click the *Delete* button to delete segment 6

14. Click *Yes* to confirm removing the segment

15. Choose *Evergreen Forest* segment. Note the area is 12237.15 acres.

16. Enter the new A*rea* = 12712.1 acres (12237.15 + 474.94). You have successfully aggregated two segments.

17. Aggregate segments by setting the *Bare Exposed Rock* segment area to 0.0, deleting it, and adding its area to the segment *Mixed Tundra*. The new total area will be 1444.1acres.

18. Choose *Done* to exit the *Edit Parameters* dialog

You are now left with 4 land segments. These are the segments for which you will now input parameters and simulate with HSPF.

20.4 Defining Land Segment Parameters

You are ready to begin defining parameters for the land segments of the HSPF model. This model will be set up for purely hydrologic analysis; thus, you will be activating and inputting parameters only to the SNOW and WATER modules of HSPF. Further, to decrease the time needed to complete the tutorial, you will fully set up one segment then copy those parameters to the other 3 in the model. Do the following to complete these tasks:

1. Select the basin icon by clicking on it

2. Select ***HSPF*** | ***Edit Parameters***

3. Choose the *Evergreen Forest* segment from the *Basin Data* window

4. Click the *Define Activities* button

5. Click the boxes next to *Snow (SNOW)* and *Water (PWAT)* to checkmark them. Note that the buttons become active upon checking the corresponding box.

You have just activated the Snow and Water modules for this land segment (*Evergreen Forest*). You must now enter the parameters for each active module to allow HSPF to simulate the segment correctly.

1. Click on the *SNOW* button in the *Pervious Land Activity* dialog

2. In the *SNOW* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values; these values are either generally recommended values or HSPF defaults):

SNOW-INIT1	SNOW-INIT2	SNOW-PARM1	SNOW-PARM2
Pack-snow = 4.0 in Pack-ice = 2.0 in Pack-watr = 2.0 in	SKYCLR = 1.0	LAT = 40.5 degrees MELEV = 8410.8 ft SHADE = 0.40	TSNOW = 35.0 degrees MWATER = 0.2

Note: these values were extracted from meteorological datasets.

1. Now that all values are entered for this segment, click the *Apply Parameters to Segments* button. This will allow you to assign these same parameters to other segments in the model.

2. Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.

3. Repeat step 2 for *Residential* and *Mixed Tundra*

4. Click *OK*. The selected segments will be assigned the same parameters input here.

The final step in entering parameters for a module in HSPF is to define time series input (External Sources) and time series output (External Targets). For the land segments in this model, you will define External Sources for input data, but no External Targets will be specified – you will choose output to be given at the outlet of the watershed, not at each land segment.

1. Click on the *External Sources* button in the *SNOW* dialog. The *Assign External Sources* dialog will appear with a list of the datasets available in the WDM file specified for this model (*littleco.wdm*).

2. If there is nothing in there (and you might also get a warning message that the file is not there), *Open* the *littleco.wdm* file and click *OK*.

3. Assign the following datasets as sources by setting the fields to the appropriate values, then clicking the *Assign* button for each line. If you make a mistake, choose the incorrect line in the lower window and click *Delete*.

Member Name	Member subscript	Units	Missing Data	Transformation	Quality flag	Multiplication Factor	Dataset
PREC		ENGL	UNDF	SAME	0	0.25	3
PREC		ENGL	UNDF	SAME	0	0.75	7
DTMPG		ENGL	UNDF	AVER	0	1.0	14
WINMOV		ENGL	UNDF	AVER	0	1.0	15
SOLRAD		ENGL	UNDF	SAME	0	1.0	16
AIRTMP		ENGL	UNDF	AVER	0	0.4	17
AIRTMP		ENGL	UNDF	AVER	0	0.6	18

1. Since you will add other External Sources to this segment with the *PWAT* module input, you will not assign the External Sources to other segments yet. Click *Done* to return to the *SNOW* dialog.

2. Click *OK* to return to the *Pervious Land Activities* dialog

You have completed the set up of the *SNOW* module. You will now repeat the same basic steps to set up the *PWAT* module for the land segments of the model.

1. Click on the *PWAT* button in the *Pervious Land Activity* dialog

2. In the *PWATER* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values):

PWAT-PARM1	PWAT-PARM2	PWAT-PARM3	PWAT-PARM4	PWAT-PARM5	PWAT-STATE1
CSNOFG = On RTOPFG = On UZFG = On	LZSN = 5.0 INFILT = 0.48 LSUR = 2258.0 SLSUR = 0.25 KVARY = 0.7 AGWRC = 0.997	DEEPFR = 0.1 PETMAX = 35.0 PETMIN = 30.0	CEPSC = 0.1 UZSN = 1.0 NSUR = 0.25 INTFW = 3.0 IRC = 0.7 LZETP = 0.8		CEPS = 0.2 UZS = 1.0 LZS = 6.0 AGWS = 1.6 GWVS = 1.9

1. Now that all values are entered for this segment, click the *Apply Parameters to Segments* button. This will allow you to assign these same parameters to other segments in the model.

2. Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.

3. Repeat step 2 for *Residential* and *Mixed Tundra*

4. Click *OK*. The selected segments will be assigned the same parameters input here.

Now you must define the additional External Sources needed for the PWATER simulation. Most of the External Sources entered for the SNOW module are also used for the PWATER module, but there is one input series that is not yet entered.

1. Click on the *External Sources* button in the *PWATER* dialog. The *Assign External Sources* dialog will appear with the sources that you created for this segment in the *SNOW* module listed in the lower window.

2. Assign the following dataset (evapotranspiration) as a source by setting the fields to the appropriate values, then clicking the *Assign* button. If you make a mistake, choose the incorrect line in the lower window and click *Delete*.

Member Name	Member subscript	Units	Missing Data	Transformation	Quality flag	Multiplication Factor	Dataset
PETINP		ENGL	UNDF	SAME	0	1.0	23

All the External Sources needed for the simulation are now assigned to this segment. You will now copy these same External Sources to all other land segments in the model.

1. Click the *Apply Sources to Segments* button. This will allow you to assign these same External Sources to other segments in the model.

2. Choose *Shrub & Brush Rangeland* in the *Available Segments* window; move it to the *Selected Segments* window by clicking the → button.

3. Repeat step 2 for *Residential* and *Mixed Tundra*

4. Click *OK*

5. Click *Done* in the Assign External Sources dialog

6. Click *OK* in the *PWATER* dialog

7. Click *OK* in the Pervious Land Activities dialog

The *Edit Parameters* dialog should now be active. You have successfully set up the *SNOW* and *PWATER* modules for all land segments in the model (remember that you set up the modules explicitly for the Evergreen Forest segment and then copied the parameters to all other segments). You may want to ensure/review the set up for the other segments by choosing the segment in the *Basin Data* window, then clicking the *Define Activities* button and reviewing the *SNOW* and *PWATER* input.

Now that the land segment data is complete, you will enter data for the reach (stream) segment of the model.

1. Click *Done* to close the *Edit Parameters* dialog

20.5 Defining Reach Segment Parameters

To simulate runoff in-stream, you will need to activate the *HYDR* module for the reach segment of the model. You will also need to specify the output dataset needed (a hydrograph in this case) to view the results of the simulation.

1. Double-click on the watershed outlet in the *Graphics Window* (this can be done while the *Edit Parameters* dialog is still present by only single-clicking the outlet). The *Reach/Reservoir Data* section of the *Edit Parameters* dialog will become active and display the name of the outlet chosen (*Little*).

2. Click on the *Define Activities* button

3. Check the box to activate the *Hydraulics (HYDR)* module of HSPF for this reach

4. Click the *HYDR* button in the *Reach/Reservoir Activities* dialog

5. In the *HYDR* dialog that appears, enter the following values in the appropriate fields (leave other fields at the default values):

HYDR-PARM1	HYDR-PARM2	HYDR-INIT
ODFVFG = 4	Length = 13.82 DeltaH = 2200.0	

The next input you must enter is the FTABLE for the reach. The FTABLE is a spreadsheet-like table that contain the conveyance parameters of the reach (depth, area, volume, and outflow). This table may be calculated manually or with the help of the *Channel Calculator* in WMS.

1. Click on the *Define FTABLE* button the *HYDR* dialog

2. In the *FTABLE* dialog that appears, enter the following values (it is possible to change the number of rows and columns in the FTABLE, but the default values will be sufficient in this case. These values are saved in a text file called *ftable.txt* in the *CH18* directory of the tutorials. You can open this file, copy the data (use *CTRL-C* to copy), and paste the data to the *FTABLE* dialog (use *CTRL-V* to paste) if you wish):

Depth (ft)	Area (ac)	Volume (ac-ft)	Outflow (cfs)
0.0	0.0	0.0	0.0
0.25	25.99	6.393	13.876
0.5	26.828	12.995	43.954
0.75	27.666	19.807	86.269
1.0	25.50	26.828	139.241
1.25	29.340	34.059	201.950
1.5	30.181	41.499	273.794
1.75	31.019	49.150	354.355
2.0	31.858	57.009	443.334
2.25	35.211	90.540	879.902

1. Click *OK* to save the FTABLE

The final task in defining the reach parameters is to specify the output you want to see from the reach and where HSPF will write the output. This is done by defining an External Target. You will specify that you want a hydrograph to be output to a dataset in the WDM (littleco.wdm) file for this reach.

1. Click on the *External Targets* button in the *HYDR* dialog

2. Set the following fields to the specified values:

Member Name	Member subscript	Units	Access	Aggregation	Multiplication Factor	Quality flag	Transformation Function
RO		ENGL	REPL	Aggregate	1.0	0	AVER

1. Select the *Use existing dataset* button

2. Click on the *Select Dataset* button

3. In the *Select Dataset* dialog, scroll down and select dataset 20 (Flow), then click the *OK* button

4. The Dataset name, type, and number should appear in the Assign External Targets dialog

5. Click *Assign* to add the External Target line to the lower window

6. Click *Done*

You have now completed the input for the HYDR module in this reach. This will be the only module active for this simulation; thus, you can exit the parameter editing dialogs.

1. Click *OK* in the *HYDR* dialog

2. Click *OK* in the Reach/Reservoir Activity dialog

3. Click *Done* in the *Edit Parameters* dialog

The input for all land and reach segments is now complete in the model. The last task before saving and running the HSPF model is to assign Mass Links.

20.6 Creating Mass Links

Mass Links control how materials (water, sediment, constituents) are transferred from land segments to reaches, and from one reach to the next. Conversions in units, such as inches/acre per day of runoff → cubic feet per second, are defined in the Mass Links also. You will enter Mass Links to transfer water from land to reach and from reach to reach for this model.

1. Select ***HSPF*** | ***Mass Link Editor***

2. When you enter the *Mass Link Editor*, it is set to begin adding/deleting mass links for PERLND segments. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
PERLND	PWATER	PERO	0.083333333	RCHRES	INFLOW	IVOL

Note: 0.0833333 is the conversion for inches/acre per day → cfs.

1. Click *Add Link* (if you make a mistake, select the link in the window and click the *Delete Link* button)

2. Change the *Segment Type* (upper left) to IMPLND

3. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
IMPLND	IWATER	SURO	0.083333333	RCHRES	INFLOW	IVOL

1. Click Add Link

2. Change the *Segment Type* (upper left) to RCHRES

3. Set the fields to the values below:

Volume Name	Volume Group	Volume Member Name	Multiplication Factor	Target Name	Target Group	Target Member Name
RCHRES	HYDR	ROVOL	1.00	RCHRES	INFLOW	IVOL

1. Click Add Link

The Mass Links needed for PERLND, IMPLND, and RCHRES segments are now set up. You are ready to save and run the model.

Click *OK* to exit the *Mass Link Editor*.

20.7 Saving and Running HSPF Simulation

The last step is to save and run the HSPF simulation.

1. Select ***HSPF*** | ***Run Simulation***

15. HSPF should run to completion. You will then be able to view the solution.

CHAPTER 21

CE-QUAL-W2 Interface

CE-QUAL-W2 is a 2-D laterally averaged hydrodynamic model.

This workshop will walk you through the CE-QUAL-W2 / WMS interface. It uses a previously generated TIN and creates CE-QUAL-W2 input files.

The required files are:

- EastCanyon.tin

21.1 Objectives

In this workshop you will get acquainted with the CE-QUAL-W2 interface in WMS. This includes the following:

7. Determining reservoir extent
8. Creating branches.
9. Creating segments.
10. System modeling

11. Initializing CE-QUAL-W2 simulation.

12. Branch identification.

13. Mapping segments to branches.

14. Editing branch/segment properties.

15. Saving CE-QUAL-W2 control input file.

16. Saving bathymetry as CE-QUAL-W2 input file.

21.2 Importing data and coordinate conversion

The following steps will teach you how to import data and make sure they are in the proper coordinate system.

4. Open up WMS, or if it is open select ***File* | *Open…***

5. Select *OK* to delete old data.

6. Select *File* | *Open…*

7. Locate and open the file "*EastCanyon.tin*"

8. Select the display options icon .

9. Unselect the Locked Vertices, the Unlocked Vertices, and the Triangles boxes.

10. Exit out of the display options dialog by selecting *OK*.

WMS coordinates are set to U.S. survey feet by default. This should be recognized every time a file is opened in WMS. If the file was saved using metric coordinates then this will need to be updated each time the file is opened. CE-QUAL-W2 uses metric coordinates in all bathymetry calculations so it is advisable to convert to metric coordinates and enter all data in meters. If the user wishes to use U.S. feet for data entry then the coordinate conversion can be performed later with the same results. A prompt in these instructions will appear later advising the user when to make the conversion.

1. Click the Coordinate Conversion option under the Edit pull down menu.

2. In the Coordinate Conversion dialog select the current coordinate system button.

3. Change the horizontal and vertical units to the appropriate system if the appropriate system is not the default value.

4. Click *OK*.

5. Change the horizontal and vertical system units to the desired system for conversion.

6. Click *OK* to exit the dialog.

21.3 Determining Reservoir Extent

Now that the TIN has been imported, the boundary of the reservoir must be defined.

4. Select the display options icon .

5. Make sure that the TIN Contours box is checked.

6. Select the Contours button to the right of the TIN Contours box.

7. In the Contour Interval section, choose Specified Values.

8. In the Contour Values dialog, enter 5500 for Contour #1, and 5720 for Contour #2.

9. Exit out of the contour values dialog by pressing OK.

10. In the Contour Method section, choose Color fill between contours.

11. Exit out of both dialogs by pressing *OK*.

What is displayed is the extent of the reservoir. Contour #1 was chosen as an elevation lower than any elevation in the reservoir. Contour #2 is the maximum water surface elevation, or the elevation at any desired stage to be modeled. By selecting the color fill between contours, the areas that fall between Contour #1 and Contour #2 have been displayed.

Once completed, the reservoir should appear as shown in Figure 1.

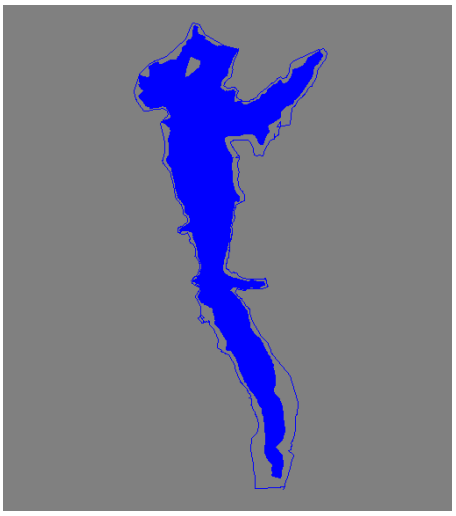


Figure 21-1: Reservoir Extent of East Canyon

There are two ways to generate the boundary of the reservoir. If you have refined your TIN such that the boundary of the TIN is the same general shape as the reservoir (as explained in the walkthrough Editing the TIN), then you can let WMS automatically generate the bounding polygon (Case 1). If not, you will need to manually create the bounding reservoir polygon (Case 2).

21.3.1 Case 1: Automatic boundary polygon generation

3. Make sure that you are in the Terrain data module .

4. In the TIN pull down menu, highlight the Conversion option and select TIN Boundary -> Feature.

21.3.2 Case 2: Boundary of Reservoir to Polygon

4. Select the Map module icon .

5. Select the create feature arc tool .

6. Trace out the outline of the reservoir at the specified water surface elevation by clicking at locations along the perimeter. Make sure that the generated polygon stays within the boundary of the TIN!

It is a good idea to create your polygon slightly larger than the boundary generated by the maximum water surface elevation. This way you ensure that the entire reservoir will be modeled. If you need to define the perimeter by defining separate arcs, double click to end the arc. Make sure and start the new arc at the end of the old one however.

7. From the Feature Objects pull down menu select Build Polygon.

8. Select the select polygon tool .

9. Click somewhere within the reservoir. If all of the arcs have been connected correctly, the entire area will become highlighted. If this does not happen, see Endnote #1.

21.4 Change Contour Display Options

For the following steps, we do not need to know the exact extent of the reservoir. Because color filled contours require more time for the image to regenerate, we will change the contour display options.

4. Select the contour display options icon .

5. Under contour interval, select Number of contours. A good value to use is 25.

6. Under contour method, choose Normal linear contours

7. Exit out of the dialog by pressing *OK*.

For WMS to process the data correctly, certain types of data must be grouped together as a coverage. In CE-QUAL-W2 modeling two types of coverages must be created, the branch and the segment.

21.5 Create Branches

Now, we are ready to start creating branches and segments. WE will start first by creating branch coverage.

21.5.1 Create Branch coverage

3. Open the folder in the upper right side of the screen labeled Map Data.

4. Right click over the coverage labeled default coverage and select the Properties option.

5. Select the pulldown menu under the Coverage Type heading and change to CE-QUAL-W2 Branch.

6. Change the elevation to 6000.0 or any elevation above the maximum elevation of the TIN.

7. Rename to Branch.

8. Exit out of the dialog by pressing *OK*.

When more than one coverage is created the current (or active) coverage appears in bold letters. Make sure that checks appear in the boxes next to all coverages that will be used"

21.5.2 Create Branches

Now that the branch coverage has been created, the next step is to define the branches.

1. Switch to the Map module .
2. Select the zoom tool .
3. Create a box around the area shown in Figure 2.

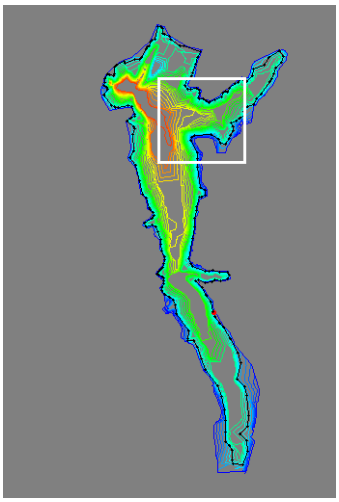


Figure 21-2: Branch area of interest

4. Select the create feature arc tool .
5. Starting at one bank (at the edge of the polygon), click to begin the arc.
6. Click on the other edge of the arc. If it has connected with the boundary polygon, it will automatically end the arc, and begin another one.
7. If the arc has already snapped to the polygon, press the escape key to end the arc.

It is possible to create a curved boundary. Although this is not common practice, it can be done by simply creating more points along the arc.

Steps 3-7 should be followed to define as many branches as desired.

8. In the Feature Objects pulldown menu, choose Build Polygon.

9. Press *OK* at the prompt.

10. Choose the select polygon tool .

11. Click within each of the sections to make sure that polygons are correctly defining the branches. If they are, then the branch should become highlighted.

If the branches are not correctly identified when selected then most likely the arcs created to define them did not snap to the boundary polygon. Using the zoom tool can help you identify these problems. If the problem persists, see Endnote #1.

21.6 Create Segments

Now, we are ready to start creating segments. We will start first by creating segment coverage. Because we have already defined the branches, we can use the branch boundaries to help define the segments.

21.6.1 Create segment coverage

1. Right click over the Branch coverage in the upper right of the screen and select the Duplicate option.

2. Right click over the segment coverage and select the Properties option.

3. Change the Coverage Type to CE-QUAL-W2 Segment.

4. Change the elevation to 6000.0 or any elevation above the high point of the TIN.

5. Rename the coverage Segment.

6. Exit out of the dialog by selecting *OK*.

7. Select the Segment coverage so that it is either highlighted or in bold.

21.6.2 Create segments

1. Select the zoom tool .
2. Create a box around the area shown in Figure 3.
3. Select the create feature arc tool .
4. Starting at one bank (at the edge of the polygon), click to begin the arc.
5. Click on the other edge of the arc. If it has connected with the boundary polygon, it will automatically end the arc, and begin another one.
6. If the arc has already snapped to the polygon, press the escape key to end the arc.

Steps 2-6 should be followed to define as many segments as desired.

A few notes about segment creation:

- The arcs used to divide the segments should be roughly perpendicular to the general flow direction
- Over the area where a branch intersects the main body, only one segment can be present (the branch can only open into one segment)

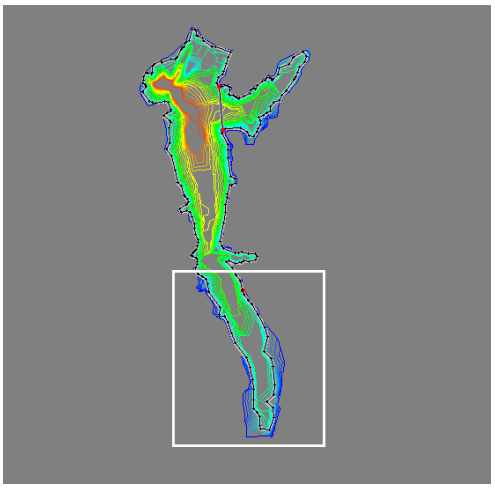


Figure 21-3: Segment area of interest

7. In the Feature Objects pull down menu, choose Build Polygon.
8. Press *OK* at the prompt.

9. Choose the select polygon tool .

10. Click within each of the segments to make sure that polygons are correctly defining the branches.

21.7 System Modeling

After the segments and the branches have been created, it is a good idea to create a conceptual model of the system. Figure 4 shows a subdivided reservoir, and its accompanying model.

Important parts of the system model include the segments in each branch, the segment that a branch enters into, and the numbering. The general trend in the numbering is that the most upstream segment in each branch has the lowest value (beginning with the segments in the main branch). All segments in a branch should be numbered sequentially. In addition, the first segment in your model should be numbered 2. This is to allow for a dummy segment required by CE-QUAL-W2 to be created as segment 1. Dummy segments are also required at the end of a branch.

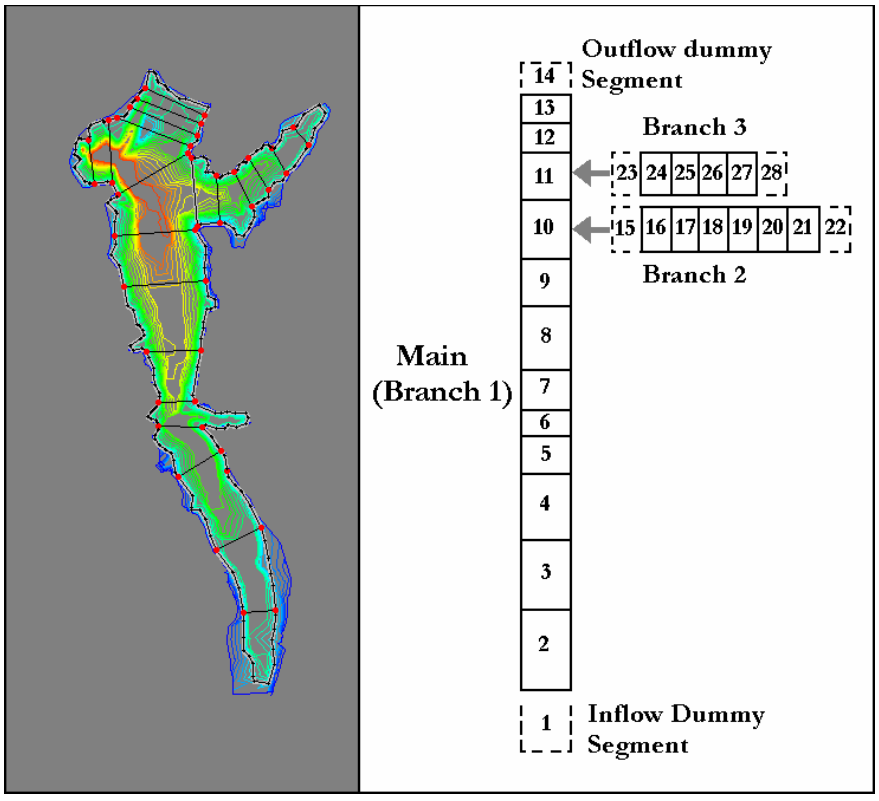


Figure 21-4: System Model

21.7.1 Initializing CE-QUAL-W2 simulation

For WMS to generate the input file, the CE-QUAL-W2 interface must be initialized.

3. In the CE-QUAL-W2 menu, choose New Simulation.

The CE-QUAL-W2 interface is now initialized.

Before beginning branch identification the coordinates will need to be converted from U.S. survey feet to meters if this was not performed at the beginning. This cannot be performed after obtaining the lengths of the branches since the measured lengths will not be converted along with the contours and polygons.

21.7.2 Branch identification

Each branch must be given its corresponding identification as created in the System Modeling step.

1. Select the select polygons tool .
2. Switch coverages to branch by highlighting the branch coverage in the upper right corner of the screen.
3. Double click on the main branch to bring up the Polygon Branch attributes.
4. Select the Initialize branch properties button if it is not already selected.
5. Give this a Branch ID of 1
6. Check the box for Main stem
7. Select the length of segment tool .
8. Click at the most upstream part of the branch.

You will generally want the length of the branch to be equal to the average length of flow that will occur in a branch. To account for a curved flowpath, you can add as many points along the line as desired.

As you trace the line through the branch it is best to click at or in each segment to provide a better representation of the orientation of the branch and it's segments. Avoid using one line to span multiple segments, rather click at each segment for better results.

9. End the line by double clicking along the most upstream part of the branch.

This will give lengths and orientation to each segment in the branch and it also maps the segments to their respective branches.

10. Follow steps 4, 5 and 7-9 for the other two branches, giving them their respective Branch ID numbers.

You may end up with more than three branches. If this is the case select branches from the CE-QUAL-W2 pull down menu and highlight the extra branch. Select the edit branch parameters button and in the following dialog select the delete branch properties button

21.7.3 Segment Numbering

Before continuing it is helpful to find the most upstream segment of each branch. In the segment coverage use the select polygons tool to find the segment ID of the most upstream segment of each branch and make a note of the ID's.

1. Switch to the branch coverage by selecting branch in the pull down menu near the top left of the screen (if it isn't already selected).
2. Select the select polygons tool .
3. Double click on the main branch.
4. Select the segments tab in the polygon branch attributes dialog and highlight the most upstream segment ID.
5. Select the make upstream segment button.
6. Repeat steps 3-5 for the other branches.
7. In the CE-QUAL-W2 pull down menu select Segment Numbering.
8. Select Yes.

This numbers the segments starting with the upstream segment as segment 2 (segment 1 is a dummy segment that is inactive). This also designates the upstream and downstream segments of each branch.

21.7.4 Editing segment properties

Within each segment, many individual properties must be defined. These include the segment length, the layer properties, and the width computations. Figure 5 shows the different parameters required for the segment.

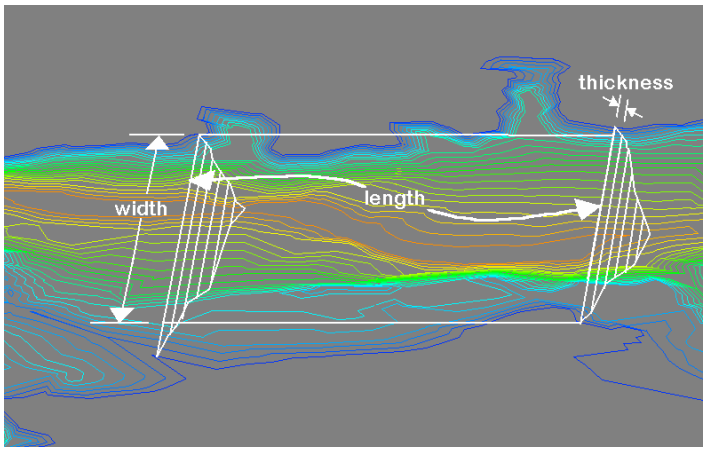


Figure 21-5: Segment bathymetry

1. Switch to the segment coverage by highlighting the segment label in the upper right corner of the screen..
2. Highlight the entire reservoir by holding the shift key and left clicking over each segment.
3. Select Layers from the CE-QUAL-W2 pulldown menu.
4. In the field labeled Top elevation enter the top elevation of the reservoir (5720).
5. Press the compute storage-capacity curve.

The computation of the storage capacity curve may take some time. A visual display is generated showing the locations and elevations that are being modeled. Figure 6 shows the CE-QUAL-W2 layer editor where the storage capacity curve is shown.

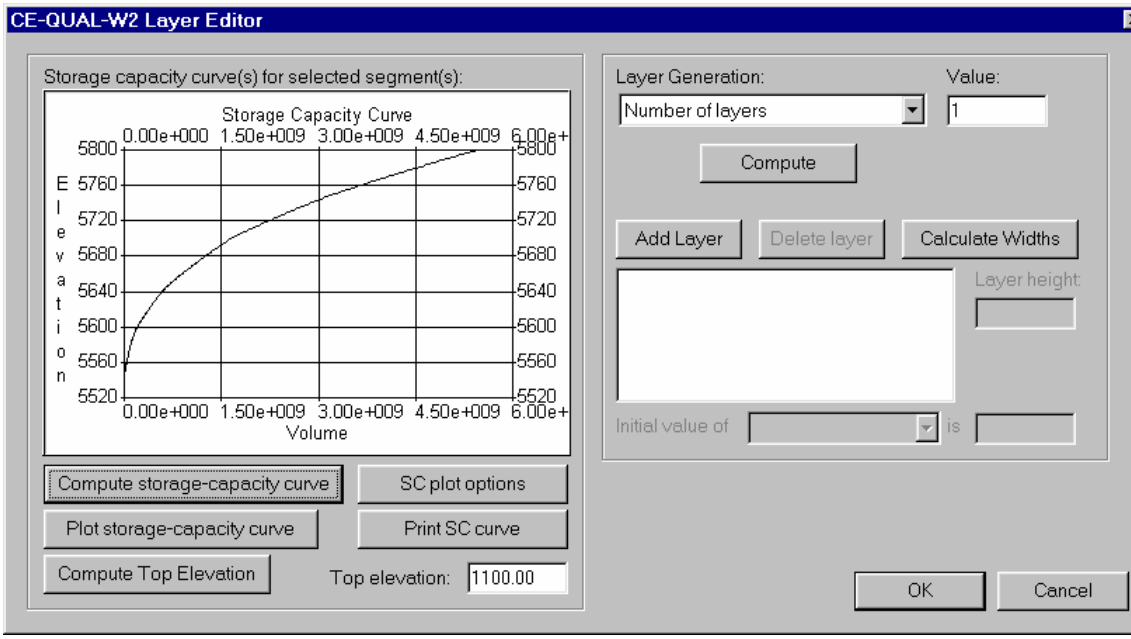


Figure 21-6: CE-QUAL-W2 layer editor

6. Select Layer Heights from the Layer Generation pulldown menu. Enter 5.00 in the field labeled Value.

Notice that you can also enter in a number of layers instead of a layer height. If the number of layers option is chosen, then the layer height will be calculated using the following equation:

H = (Max elevation on TIN – Min elevation on TIN) / Number of layers

If the layer height option is used, then all of the layers will be that specified thickness, except for the top layer, which will be a remainder of what is left.

7. Press the calculate widths button. If a prompt telling you that 'Some of the layers are below the minimum elevations in the storage capacity curve', press OK.

8. Exit out of the layer editor by pressing *OK*.

WMS computes the widths of the segments using the layer height, length, and the volume computed from the storage capacity curve. In order for a layer to have a zero width, then its volume must be zero, meaning that the minimum elevation of the segment lies above the maximum elevation of the layer.

9. Exit out of the layer editor by pressing *OK*.

21.8 Saving bathymetry as CE-QUAL-W2 input file

11. 1. In the CE-QUAL-W2 menu, choose Save simulation.

12. 2. Find an appropriate folder, and save the simulation there.

In saving the simulation, many files are created. However, the main file of interest is the bathymetry input file, bath.npt. At this point, the other files do not contain any useful information.

Since bath.npt is the only file of significance, it is recommended that it be moved to a folder that will eventually contain all of the final CE-QUAL-W2 input data. The other files can be deleted at this point.

This chapter is not complete.